C&R TECHNOLOGIES®

CRTech TD Direct®
User's Guide

Instructions for TD Direct
and Introduction to SpaceClaim Engineer®

For C&R Thermal Desktop® Version 5.8
July 2015

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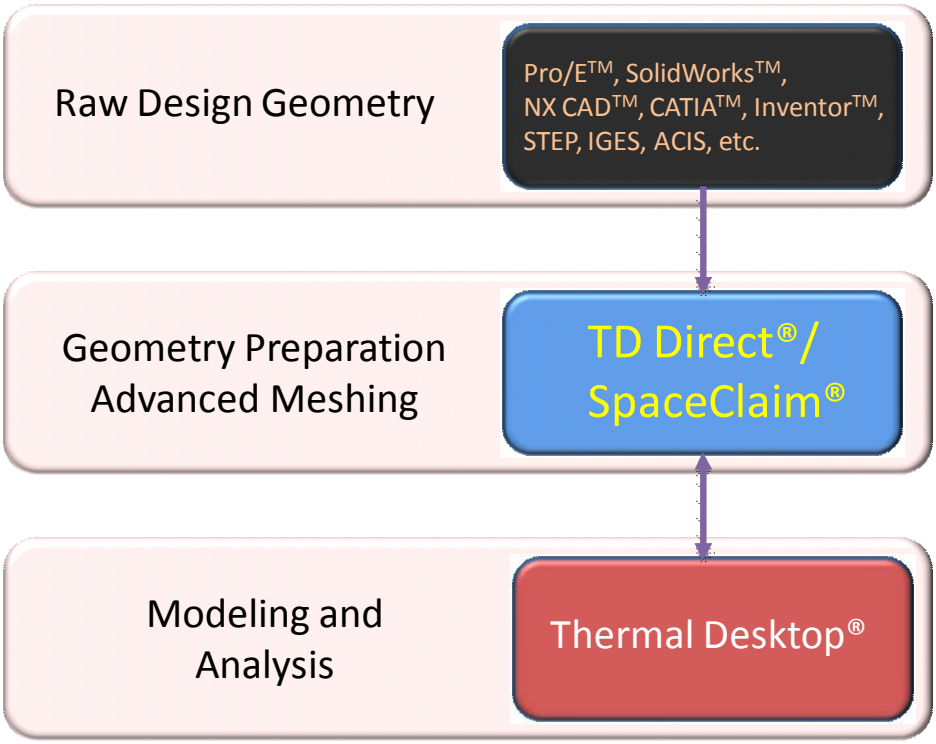
1 Introduction

This document introduces CRTech TD Direct®, a tool for generating C&R Thermal Desktop® models directly from CAD geometry. Geometry may be imported from external systems such as Pro/E, SolidWorks, Catia, and NX, as well as STEP and IGES formats. Solid and surface geometry may also be created natively using simple and intuitive direct modeling features.

TD Direct is CRTech's extension of SpaceClaim Engineer®. Together, they allow geometry creation, importation, defeaturing, healing, and simplification of detailed design models in preparation for thermal model building. TD Direct provides powerful finite element meshing with an easy to use thermally oriented user interface that directly and dynamically links with a Thermal Desktop model.



For sake of simplicity in this document, the use of the name TD Direct implies both the functionality provided by CRTech's extension and the underlying functionality provided by SpaceClaim Engineer. In general, the geometric creation and modification functionality is provided by SpaceClaim Engineer, and the thermal specific editing and mesh generation are provided by TD Direct. The relationship between TD Direct and SpaceClaim Engineer is identical to the relationship between Thermal Desktop and Autodesk's AutoCAD®. TD Direct is seamlessly integrated with SpaceClaim Engineer, and together they present a single coherent user interface for thermal model preparation and construction.

TD Direct is automatically installed by the Thermal Desktop installer. If you have the latest Thermal Desktop version installed, you will also have the latest TD Direct version installed. TD Direct can be installed (via the Thermal Desktop installer) before or after SpaceClaim Engineer is installed. Once installed and licensed, the TD Direct extensions will appear when SpaceClaim Engineer is launched. SpaceClaim Engineer can be obtained from CRTech as a bundle with a license for TD Direct, or an existing SpaceClaim Engineer installation may be used.

Model geometry can come from many sources (customers, co-workers, suppliers) and also in various formats. Conversion or incorporation of this data into a thermal/fluid model must often be repeated as designs are revised. TD Direct maintains a live, bidirectional link with Thermal Desktop for model synchronization. Geometry in TD Direct can be parameterized and driven by Thermal Desktop symbols, automatically updating the finite element mesh and connectivity information with a one-button synchronization operation.

Thermal model information is applied directly to the high level CAD geometry, and when any changes to dimensions, material properties, optical surface treatments, submodels, etc. are made, a new thermal model is just one button click away. The synchronizing operation exchanges data in both directions between Thermal Desktop and TD Direct. TD Direct is synchronized with Thermal Desktop information such as submodels, material properties, optical properties, and radiation analysis groups. Thermal Desktop is synchronized with a new thermal model and geometry. In effect, TD Direct is an extension of the Thermal Desktop user interface, directly linking CAD data to a thermal model.

TD Direct allows the use of FEM and FD methods simultaneously. Using a new concept called *Domains* (covered extensively later in this document), connections between various components of a model are retained when any of the components are updated. TD Direct allows components generated from CAD data to be added to an existing FD based thermal model. Any aspect of these components may be changed, without having to respecify connections to other components.

Advances in CAD technology are making it easier for non-specialists to use these programs, even if their use is sporadic: ease-of-use provided by direct modeling is significantly flattening the learning curve. Increasingly, the thermal engineer is not just the passive recipient of model geometry, but also the active generator of geometric models. Even if thermal engineers don’t originate the design, knowledge of how to adjust and change it, or make quick models of simplified equivalent parts, ground planes, test chambers, etc. is necessary.

Direct modeling methods provided by SpaceClaim are a revolution in ease-of-use for creating and manipulating CAD geometry. Faces and edges are selected and “pulled” to modify the geometry. The powerful direct modeling algorithms infer the relationships between the geometry and make appropriate modifications globally, without the need for a history tree.

TD Direct allows the user to quickly generate geometry natively, and/or modify, defeature, and simplify CAD geometry imported from a variety of formats, and then use that geometry for FE mesh generation or to assist FD modeling. The advanced features of these tools along with a dynamic, real-time coupling with Thermal Desktop allows thermal models to be automatically updated when changes to the design occur.

1.1 Purpose

TD Direct should be used when:

- The geometry is complex and cannot be easily modeled with Thermal Desktop finite difference surfaces and solids. This includes the complexities associated with contacting surfaces and solids.
- Frequent and wholesale updates to the geometry are expected, in which grip point stretching and shrinking of Thermal Desktop finite difference entities may not be enough to maintain fidelity to the design or become too time consuming.
- The geometry has been developed in other CAD tools.

Of course, a system-level model in Thermal Desktop can be composed of many parts and assemblies, so some portions may use one type of modeling (such as native FD surfaces and solids) while other portions use imported objects. A Thermal Desktop drawing can link with multiple TD Direct documents for modular model building.

This manual assumes that the user is already very familiar with basic Thermal Desktop modeling, as covered in the separate Thermal Desktop User's Manual.

1.2 Important Concepts

This section represents a brief summary for reference only. Explaining these concepts is a primary purpose of the rest of this document.

Geometric Model vs. Thermal Model. The first generation of thermal engineering software divided the world up into two parts: a radiation tool (e.g., TRASYS), and a tool to compute temperatures (e.g., SINDA). The radiation tool computed radiation exchange factors and orbital heating loads. These were the boundary and matrix terms for the temperature solver. The input to the radiation tool consisted of an input file that described surfaces such as rectangles, discs, and spheres. This was often called the "Geometric Math Model," or GMM, or just "geometric model." The set of conduction terms, radiation terms, and boundary conditions that was input to the temperature solver was called the "Thermal Math Model," or TMM, or just "thermal model." The term "geometry" became synonymous with radiation, since the rest of the thermal model was generated by hand. Although such a division is an anachronism, the terminology still persists and can be a source of confusion.

The situation is much more complex today, since the term "geometry" can have many meanings. Geometry is not just used to calculate radiation, it is also used to generate conduction and capacitance terms, and to represent even higher level objects such as FloCAD pipes that compute fluid flow, convective ties, radiation, conduction, and capacitance terms.

With the advent of Thermal Desktop and its integration with AutoCAD, geometry used for design and manufacturing was also made available for use in thermal analysis. This approach is taken a step further with the integration of TD Direct and Thermal Desktop. Now, design geometry from many different sources can be imported, manipulated, and used for generating thermal models. Thus, there is design geometry created by CAD systems for the purposes of manufacturing, simplifications of the detailed design represented in CAD systems intended for use in engineering analysis, and native Thermal Desktop analysis geometry used to compute both radiation and conduction effects.

The concepts related to geometry and thermal models will be expanded in later sections, but it is helpful to abandon the traditional definitions of "thermal math model" and "geometry." The thermal model consists of many components, including geometry. Geometry has many sources, and where ambiguous, will be prefaced with a description. For example, "design geometry" is used to denote geometry constructed by a CAD system for the purpose of design and manufacture, and "analysis geometry" refers to geometric entities that are directly a part of the thermal model, such as a finite element or Thermal Desktop cylinder. Finally, for lack of a better term, "engineering geometry" will be used to refer to a CAD (or CAE if you prefer) representation of the design, simplified and abstracted to the appropriate level in preparation for analysis.

TD Direct is used to create engineering geometry, either interactively with native commands or by simplifying an imported detailed design. TD Direct provides a Thermal Desktop-like interface to assign thermal properties to the engineering model. When synchronized with Thermal Desktop, TD Direct computes a finite element mesh from the engineering model and automatically updates the associated Thermal Desktop drawing with the new mesh.

TD Direct *Domains*. Any edge, face, or solid in TD Direct can be assigned to one or more "domains." (For 2D surfaces, each side can be separately assigned.) Domains are a general-purpose identification technique. What is *really* being named is the collection of finite element vertices (nodes), surfaces, edges, and solids that will result from that geometric entity. Those thermal modeling elements don't yet exist (or might be changed later), so *domains* provide a way to refer to them indirectly.

Each domain will generate appropriate Domain Tag Sets (see below) when the object is meshed (or is remeshed) and the mesh is sent to Thermal Desktop. For example, placing the top square of a cube in the domain "upmost" will result in Thermal Desktop domain tag sets "upmost_nodes" and "upmost_surfaces," which can be used for contact, conductance, ties, and automating post-mesh editing operations.

TD Direct *Tags*. "Domains" are one type of "Tag," which is a TD Direct mark-up of an edge, surface, or solid. TD Direct geometry entities might also be assigned material properties, submodel designations, insulation specifications, and localized mesh controls. Any such application of a piece of information to an edge, surface, or solid is generically referred to as a "tag." The current set of tags may be viewed in the **Tag Tree**.

All of these Tags (domains, thermal properties, mesh controls etc.) are remembered when the geometry is reshaped.

Thermal Desktop *Domain Tag Sets*. Domain Tag Sets are like AutoCAD groups: a placeholder for referring to underlying members of a set. Nodes, surfaces, etc. can be placed in domain tag sets and referenced by other Thermal Desktop objects. The set of objects acted upon by a Thermal Desktop object (e.g., a heat load), is called the object's *applied domain*.

The applied domain can consist of directly specified objects (e.g. a surface picked from the graphics area) as well as named references to Domain Tag Sets. When a Domain Tag Set reference is used (e.g., "top_side_nodes"), the heat load is applied to whatever entities that domain tag set currently contains as members. The contents of a domain tag set can change, automatically changing the applied domain of the TD objects that use them.

Conductors, contactors, ties, heat loads, etc. can be established using *directly* specified objects by selecting those objects in the TD graphics area. Other thermal entities can also be established with *indirectly* specified objects by using the name of a domain tag set. The primary advantage of this indirect method is that the conductor, contactor, tie, or heat load does not change (or get deleted) if the members of a domain tag set are added or deleted, even if the domain tag set is empty. Whatever is defined in the domain tag set at the time an analysis is performed is what is used.

A primary purpose of TD Direct domain tags are to automatically generate Thermal Desktop domain tag sets. For example, if the TD Direct surface on the bottom of a box is placed in a domain named "mount_side", two Thermal Desktop domain tag sets will automatically be created: "mount_side_nodes" and "mount_side_surfaces". A contactor can be established between a native Thermal Desktop surface as the "from" entity, and the automatically created domain tag set "mount_side_surfaces" as the "to" entities. The box can be reshaped, renodalized, or even deleted and the contactor will persist, and contact will be re-established based on whatever the current members of the domain tag set are at run time. This key topic will be explored in depth in later sections.

1.3 Software Modules

The following software modules are referenced in this document:

SINDA/FLUINT. CRTech's core batch-style solver of thermal/fluid design or simulation problems posed as networks, where "networks" can represent finite element models (FEM), finite difference models (FDM), or both. The focus in this manual is on the SINDA (thermal network) side of SINDA/FLUINT.

Thermal Desktop. Thermal Desktop ("TD") is CRTech's geometry-based model development tool for SINDA/FLUINT modeling. The TD core module can be extended with either or both of the RadCAD module (for thermal radiation heat transfer) or the FloCAD module (for thermohydraulics and heat pipes). In this manual, FloCAD will only be rarely mentioned, and the distinction between TD and RadCAD will not be emphasized

Autodesk AutoCAD. Thermal Desktop is based in Autodesk's AutoCAD product. As such, any capability of AutoCAD is also available to the TD user. *The user is assumed to be using AutoCAD 2012 or later.*

SpaceClaim Engineer. SpaceClaim Engineer is SpaceClaim Corporation's popular direct modeling CAD program. It is a complementary tool to Thermal Desktop, especially designed for engineers to provide geometric model preparation (including import, healing, and defeaturing). In this manual, this product may simply be called "SpaceClaim." The user is assumed to be using SpaceClaim 2012 or later, and is highly encouraged to update to the latest version available.

TD Direct. CRTech's extension to SpaceClaim that assists in engineering model preparation and advanced meshing. TD Direct and SpaceClaim exist as a single executable module, much like Thermal Desktop and AutoCAD do. Since TD Direct is an extension to SpaceClaim, the use of the term "TD Direct" implies the features and functionality of SpaceClaim.

1.4 What's New

1.4.1 Version 5.8

With the release of Version 5.8, the following new features have been added:

1. The 2.5D Mesh has been changed to "Reduce Thermal Dimension." Previously only available from surface to surface across a solid, it is now available from edge to edge across a surface.

1.4.2 Version 5.7

With the release of Version 5.7, the following new features have been added:

1. The Quality of meshes can now be controlled by the user (See "Mesh Quality" on page 4-16).

2. TD Direct has powerful solid editing capability, but in many cases users only need to create a mesh of the surfaces. A Surface Mesh option was added that will mesh the faces of solids without meshing the underlying solids (See "Surface Mesh" on page 4-7).

3. Curved Elements are now available. Curved elements can use fewer nodes to represent the geometry (See "Curve Mesh" on page 4-6).

4. Measurement Symbols allow passing location of a point, length of an edge, area of a surface, or volume of a solid to Thermal Desktop as a symbol (See "Measurement Symbols" on page 4-36).

5. Grip Manipulators (See Thermal Desktop User's Guide) can be created on vertices in TD Direct (See "Grip Manipulators" on page 4-37).

1.4.3 Version 5.6

New for Version 5.6 is a renaming of the product from *CRTech SpaceClaim* to *CRTech TD Direct*. CRTech SpaceClaim existed as a bundle of SpaceClaim Engineer and a custom CRTech extension to provide the dynamic link with Thermal Desktop (internally known as Thermal Adaptor). It had a separately licensed module for finite mesh generation named "Mesh Generation for SpaceClaim." TD Direct combines the dynamic link to Thermal Desktop and the mesh generation option with the features of CRTech SpaceClaim, but is provided as a separate product from SpaceClaim Engineer. TD Direct requires SpaceClaim Engineer, in the same manner that Thermal Desktop requires AutoCAD. TD Direct may be provided with SpaceClaim Engineer from CRTech, or you may use an existing installation of SpaceClaim Engineer.

TD Direct + SpaceClaim Engineer =

CRTech SpaceClaim + Mesh Generation for SpaceClaim

TD Direct now provides the dynamic link with Thermal Desktop as well as an advanced, thermal-oriented finite element mesher as a single product.

In addition, several enhancements have been added from the Version 5.5 release:

1. Material orienters for complex shapes may be specified by using surface normals and edges. Elements generated by TD Direct will automatically have a local material orienter computed from the high level geometry.

2. Layers defined in TD Direct will be exported to Thermal Desktop. The layer color defined in TD Direct for the layer will also be exported to TD.

3. Submodel, material property, optical property, and radiation analysis groups defined in a linked Thermal Desktop drawing will appear in the pull-down menus for TD Direct editing operations.

4. Surface normals of the geometry in TD Direct are used to orient the node number display in Thermal Desktop.

5. Structured meshing is now provided. Edges may be swept across a face to produce a regular array of quadrilateral elements, and faces can be swept across solids to generate a regular array of solid hexahedral or wedge elements.

6. 2.5 D meshing is now provided. - Thin solids may be generated with a structured swept mesh of one layer, but have identical node numbers on both sides of the solid. Through thickness effects are automatically eliminated in the SINDA/FLUINT network providing the efficiency of a 2D network, but the geometry remains 3D for purposes of contact with other portions of the model. This often eliminates midsurfacing problems, such as cases where the sides of a solid no longer make

contact with other objects after having been reduced to a midsurface.

7. Improved display of problem areas. - Regions that contain geometric defects (which may be healed using TD Direct) that prevent meshing are highlighted with an improved display.

8. Tag tree generation is suppressed for SpaceClaim Engineer documents that do not contain any thermal analysis information. Previously, all entities in a document were searched for thermal information to display in the Tag Tree. This would slow the loading for very large detailed design models.

9. “Do not generate mesh for this entity” feature deprecated on the Advanced Local tab of the Mesh Control Edit form. Use external components instead.

2 Getting Started

This section describes how to establish links between one or more TD Direct documents and a Thermal Desktop model. It is an overview of how the two programs work as a unit and how to set up that relationship. More information on how to create, import and modify geometry in a TD Direct/SpaceClaim session is presented in Section 3. Instructions on how to assign information to the geometry for creating a thermal model is presented in Section 4.

A link is established by creating a *TD Direct importer* as part of a Thermal Desktop model. This importer establishes communication with a TD Direct document. Once a link is established, data is automatically synchronized between the two programs so that they function as a single integrated application.

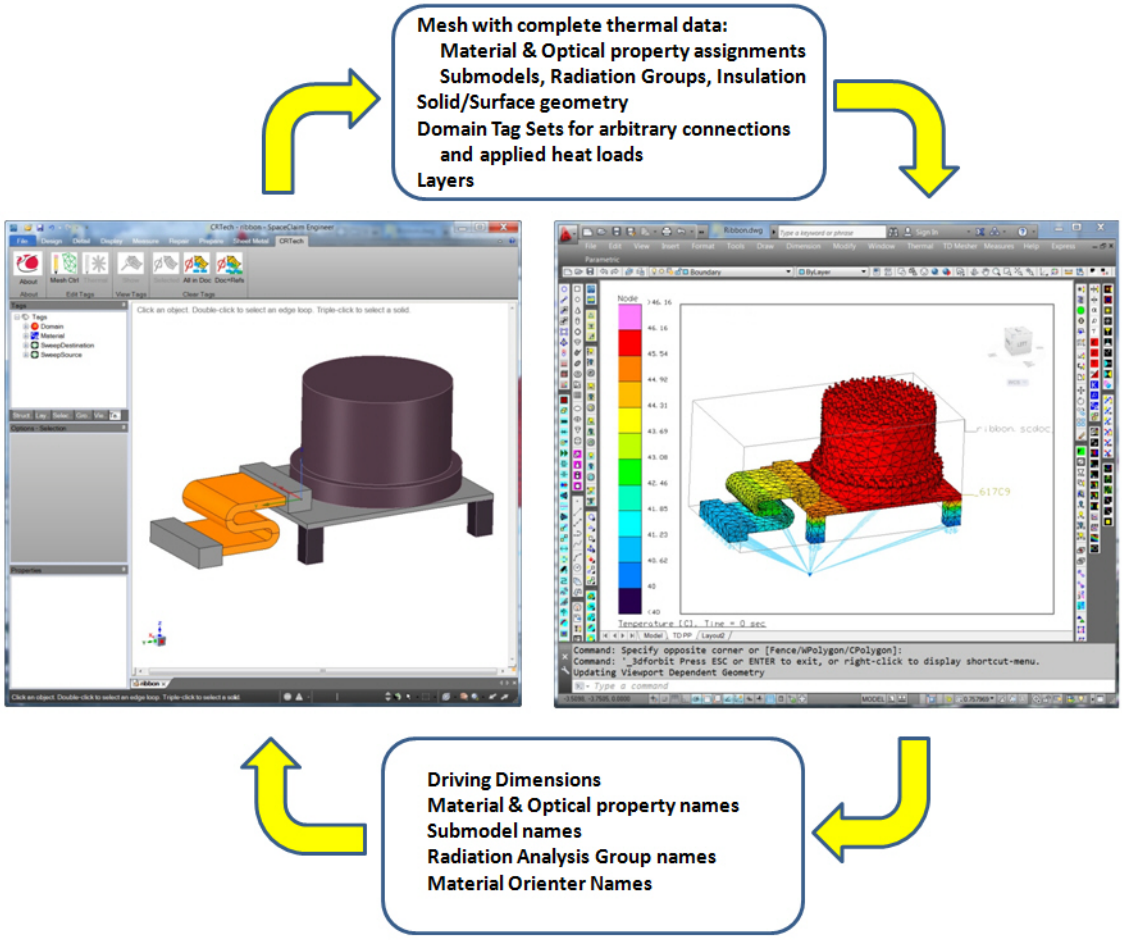


Figure 2-1 TD Direct and Thermal Desktop synchronize thermal model data to work together as a single application

The data files associated with TD Direct have the extension *.scdoc, and are referred to as *documents*. The data files associated with Thermal Desktop have the extension *.dwg and are referred to as *drawings*. A TD Direct importer created in a Thermal Desktop drawing

links to a user-specified TD Direct document. Subsequent sections will describe how to create and modify imported CAD geometry in TD Direct, as well as how to assign thermal model information for meshing. This section focuses on establishing the dynamic link between the two programs.

The analysis process will typically begin by creating a TD Direct document, and then sketching or importing CAD design geometry into the TD Direct document. After geometry is prepared, thermal model data is assigned to the geometry in TD Direct using an interface that is organized much like Thermal Desktop. Work can continue solely in TD Direct up to the point that the meshed model needs to be imported into Thermal Desktop. However, establishing a link earlier in the analysis process will have some benefits.

Establishing a link with the Thermal Desktop drawing file after geometry creation, but before beginning the thermal model building steps in TD Direct will supply the TD Direct document with existing thermal information that is present in the Thermal Desktop drawing. Existing material property, optical property, submodel, radiation analysis group, and material orienter names will appear on the TD Direct drop down menus when assigning thermal model data to geometric entities.

After creating a TD Direct document and either importing or creating some basic geometry, establishing the link to Thermal Desktop also allows the placement of the TD Direct geometry relative to the Thermal Desktop model coordinate system. If the entire model is being constructed in TD Direct, it is not necessary to reposition the importer. However, one or more TD Direct documents may be linked to a single thermal model (which may contain native FD entities), and each may be placed as desired.

After the link between the TD Direct document and the Thermal Desktop model is established, work will continue in both applications to complete the analysis process. TD Direct will be used to assign mesh controls and thermal model data. Thermal Desktop will be used to view the mesh, define connections to the rest of the model, and compute results. *Synchronization* is the operation that transfers data between the two programs and is done automatically after editing the link in Thermal Desktop. All links to TD Direct documents in a Thermal Desktop model can also be synchronized with a single button click from the tool bar or ribbon.

2.1 Creating a TD Direct Document

When TD Direct is installed, a SpaceClaim document also becomes a TD Direct document. Thermal model specific information is stored as part of the SpaceClaim document. Launch SpaceClaim by using the Start menu, or the icon placed on the desktop by the SpaceClaim installer. An empty document will be created with the default name *Design1.scdoc*. A blank document will appear as shown in Figure 2-2.

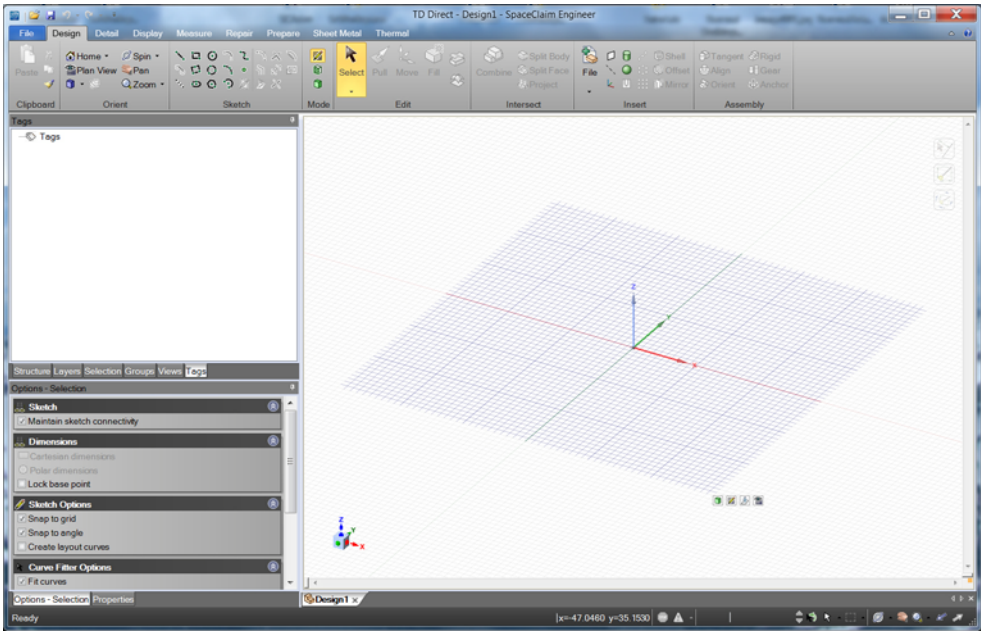


Figure 2-2 TD Direct/SpaceClaim session at startup

The appearance of the main window may vary depending on the chosen default appearance style, and the top ribbon bar may appear as in Figure 2-3. A circular logo was used in earlier versions of SpaceClaim instead of the File tab. Selecting either the File tab or the circular logo in the upper left hand corner will produce the form shown in Figure 2-3. Select Save As to save the document to a desired working directory with a more meaningful name.

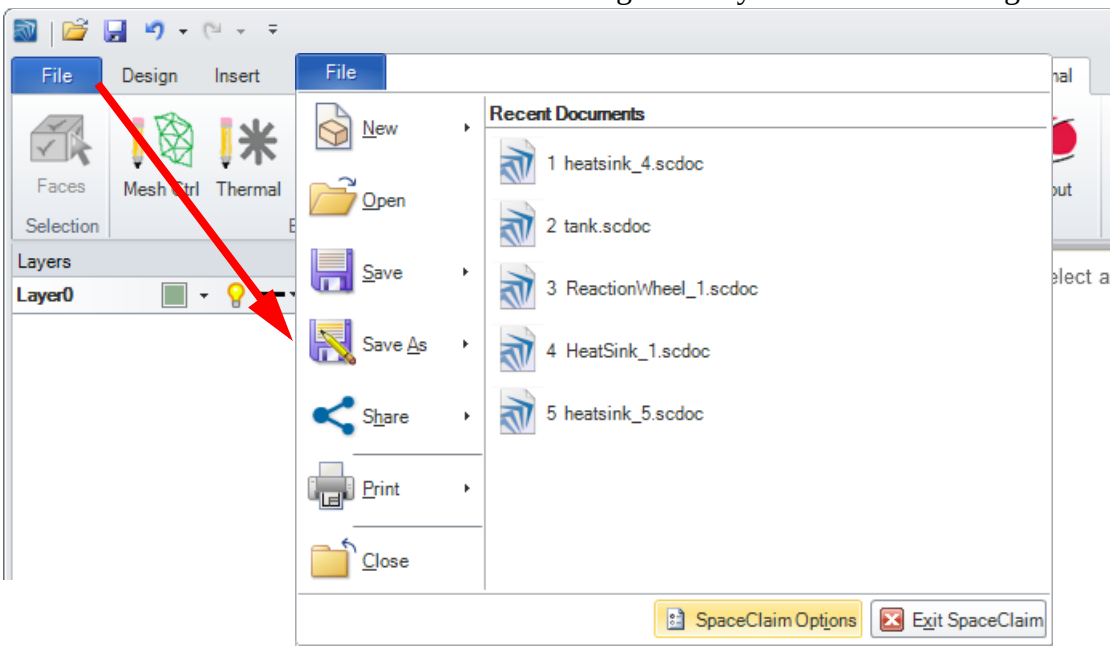


Figure 2-3 Earlier versions of SpaceClaim used a round icon instead of the File menu

When starting TD Direct for the first time, an option must be set to allow Thermal Desktop to communicate properly. Select the SpaceClaim options button at the bottom of the File form to display the SpaceClaim options as shown in Figure 2-4. On the left hand side, select File Options > General. Make sure that Load model in background is unchecked. This is a saved global option and only needs to be set once on your computer.

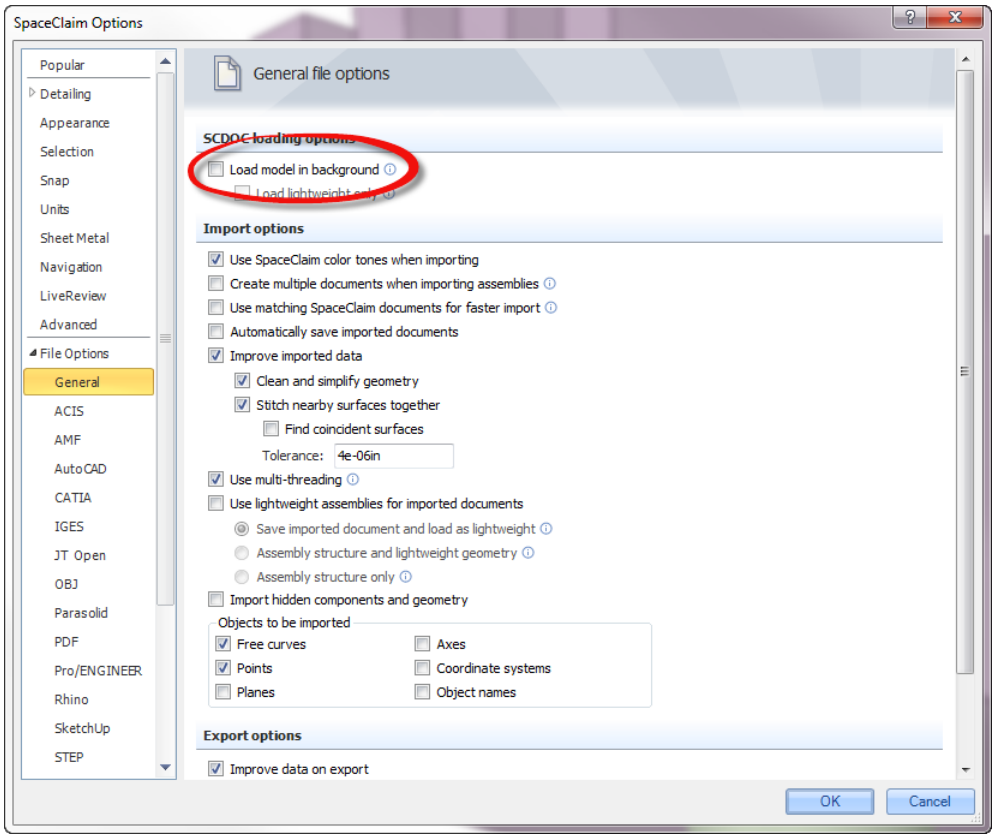


Figure 2-4 SpaceClaim options form

Common uses of the options form are to set units for length (Units), to control the appearance of the ribbon bar (Appearance), and to check for updates to SpaceClaim (Resources). A common practice is to set options and other defaults as desired, and save the document as a template document on your desktop (Any name may be used, for example, *thermal.scdoc*). Instead of launching SpaceClaim each time and doing File->Save As, simply copy the template document to your working directory. Rename the file and double click to launch TD Direct/SpaceClaim.

Geometry is created natively and imported using the Design tab. The Sketch group is used to create curves on a specified sketch plane. These curves are pulled into solid and surface shapes using the tools in the Edit group, particularly the Pull tool. CAD geometry can be imported from virtually any source using the File command on the Insert group. These tools are shown in Figure 2-5. SpaceClaim contains a revolutionary and easy to use interface, but it also contains many powerful features. It is well worth the time spent to

become proficient with the operation of SpaceClaim and the user is urged to take advantage of the tutorials and videos described in Section 3 "Creating, Importing, and Preparing Geometry".

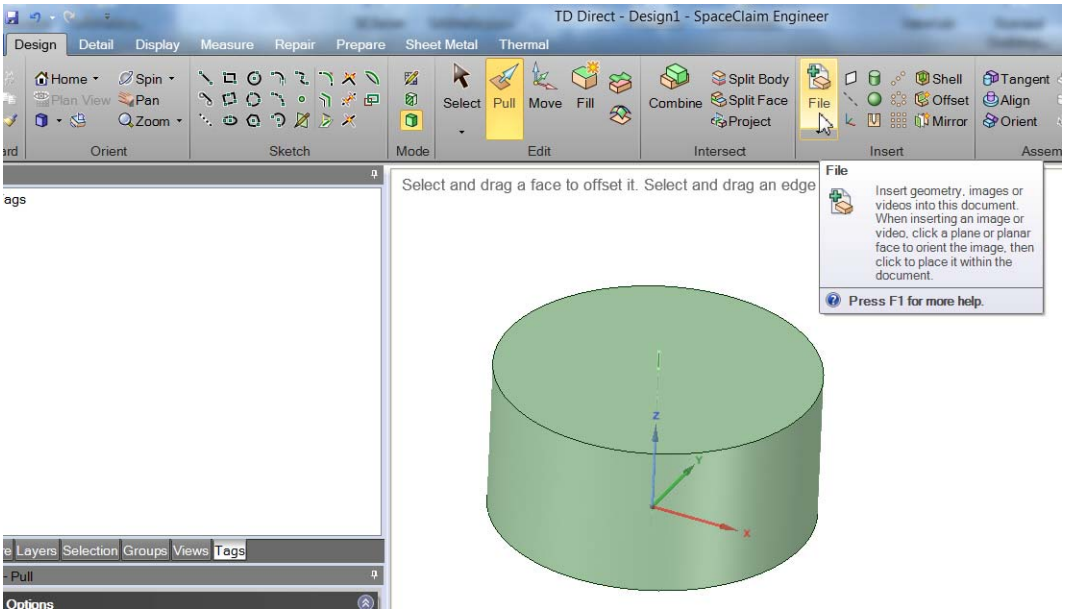


Figure 2-5 Tools for creating and importing geometry are found on the Design tab

TD Direct provides special user interface elements, directly integrated within SpaceClaim. The Tag tab on the SpaceClaim Structure pane (which also contains tabs for the structure, groups, selection, layers, and views) shows the thermal and mesh control information that is applied to the geometric entities in the document. The Thermal tab on the main ribbon contains editing tools that allow this information to be applied to the geometric entities in the TD Direct document. Figure Figure 2-6 shows thermal information being applied to the highlighted surface displayed in the SpaceClaim canvas area. Mesh controls can also be assigned globally and locally to selected entities using tools in the Thermal tab. Section 4 "Meshing with TD Direct" provides a complete explanation of the features of the Thermal tab.

2.2 Creating a TD Direct Importer

A TD Direct importer is a Thermal Desktop object that controls the geometry and/or mesh imported into Thermal Desktop from a TD Direct session. The importer can be repositioned within Thermal Desktop to orient the geometry within the thermal model as desired. A Thermal Desktop model can have multiple TD Direct importers. Each importer should reference a unique TD Direct document. If multiple copies of a part are desired in the thermal model, copies can be made by constructing multiple TD Direct assemblies, each in its own

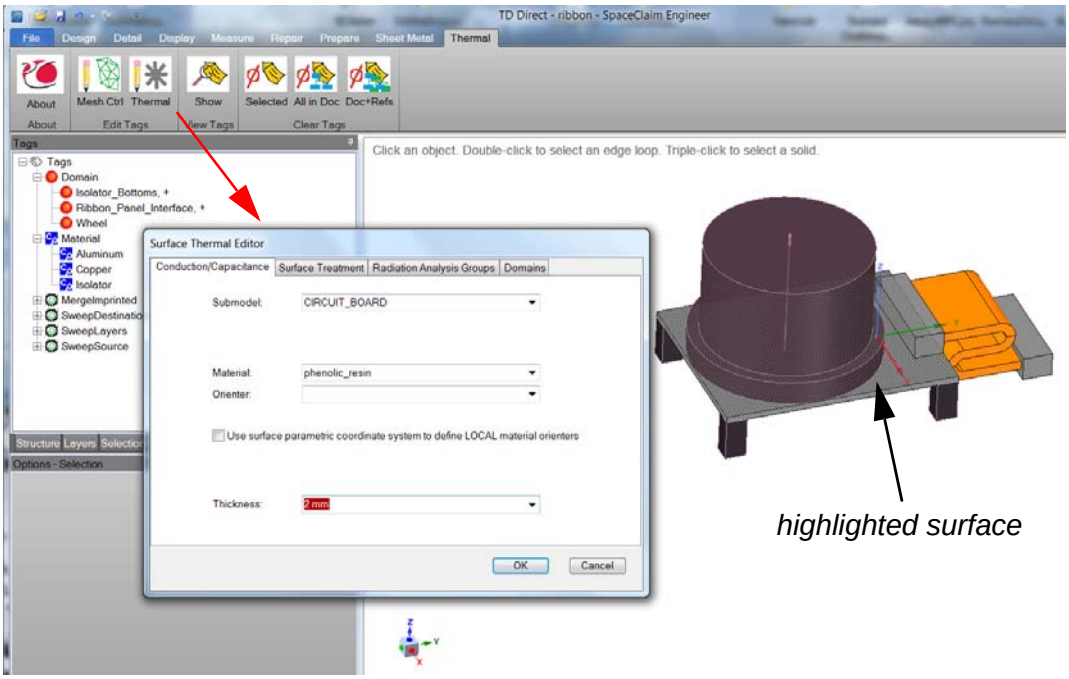


Figure 2-6 The Thermal tab provides functions for assigning thermal model data and both global and local mesh controls

document, or by constructing a master TD Direct document that instantiates multiple copies. Multiple copies and assemblies are covered in detail in Section 4.5 "Assemblies and Multiple Instances".

A TD Direct importer may bring CAD geometry and/or a finite element mesh from the TD Direct session. It is often a good idea to import the geometry first for verification of proper position in the TD drawing before generating a mesh. Geometry may also be imported without being meshed to assist in FD primitive creation by providing key points for snapping operations.

In Thermal Desktop, select Thermal > TD Direct > Create Link to create a TD Direct importer. When this is done, a File Open window will appear with the filenames filtered to *.scdoc files (the TD Direct/SpaceClaim file extension). After selecting the desired document, the TD Direct Importer window will open (Figure 2-7).

The functions of each of the controls on the TD Direct Importer form's tabs will be discussed in detail in Section 2.3. This section focuses on an overview of working with the TD Direct importer.

After options are specified, select Synchronize to update the geometry in Thermal Desktop to match the geometry in the TD Direct document. Data from Thermal Desktop is also sent to TD Direct when synchronizing. When synchronizing for the first time with a newly created importer, simply select Synchronize with the default options to get started.

The graphical representation of the importer is a bounding box around the imported geometry and mesh displayer (if a mesh is generated). The document name is also attached to the right of the bounding box as seen in Figure 2-8. An internal identifier (called the AutoCAD handle) is also appended at the end of the document name. To edit the importer,

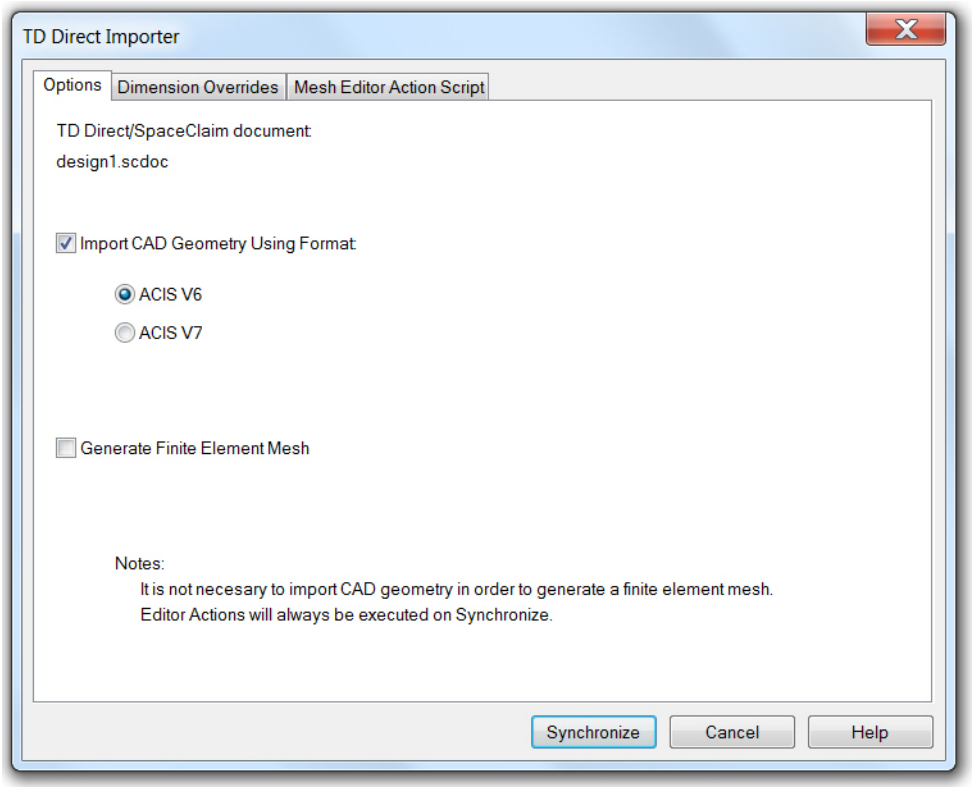


Figure 2-7 TD Direct Importer editing form

select the bounding box or the attached document name. Then select Thermal > Edit to access the TD Direct Importer form to make changes to the import options, driving dimensions, and editor actions, or to simply update the geometry or mesh from changes in the TD Direct document. A TD Direct importer may also be edited from the Model Browser.

The bounding box may also be selected to perform a move or rotate on the importer. When the importer is moved or rotated, it will also move and rotate the CAD geometry and the mesh. The mesh displayer is a native Thermal Desktop object and when managed by a TD Direct importer cannot be repositioned by selecting it directly. It must be moved using the importer. The CAD geometry imported from TD Direct is created as native AutoCAD objects and as such it is possible to reposition them by selecting directly. However, when the importer is synchronized, the CAD geometry will be reset to the transformation supplied by the importer. *Always use the importer to relocate the mesh and the geometry.* Use the importer bounding box to select all items to be attached to an assembly or tracker.

2.2.1 Controlling Visibility

The TD Direct importer manages two main items: the imported CAD geometry and the mesh. The mesh displayer in turn manages the nodes, surface elements and solid elements that are created by TD Direct. These items all occupy the same spatial locations and displaying all items simultaneously causes visual clutter as the items attempt to draw over each other.

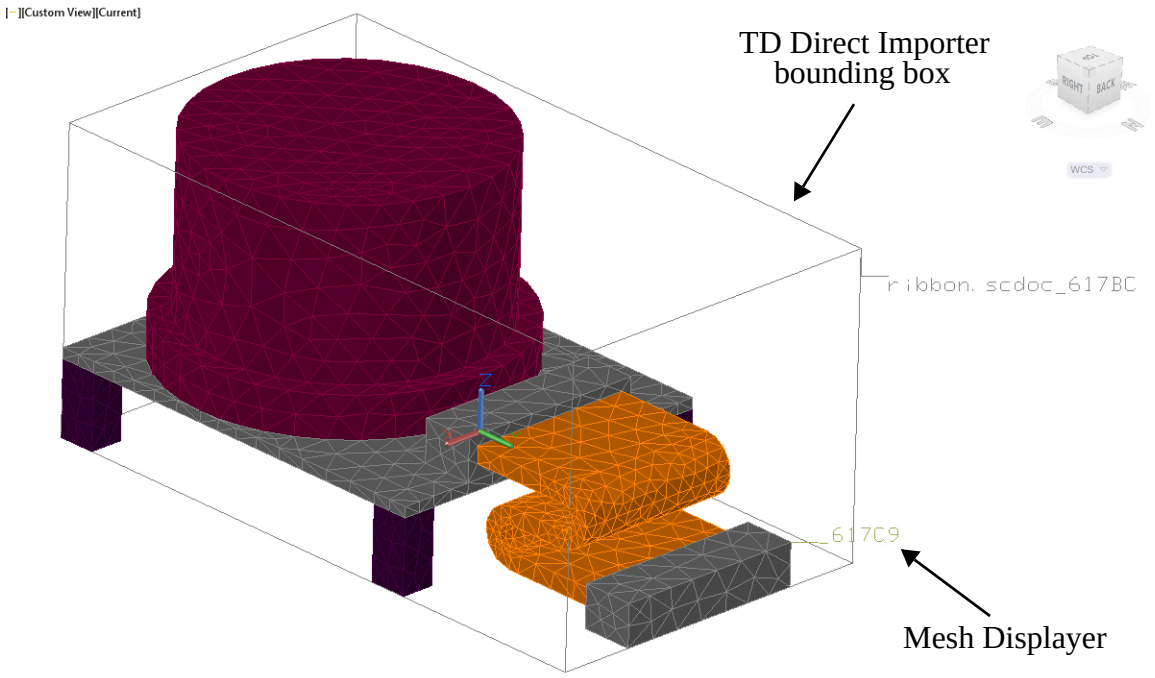


Figure 2-8 TD Direct importer may be edited by selecting its bounding box

To avoid this, the initial visibility of each of the items is set depending on what is being imported. The visibility of all items can be changed and are retained for subsequent synchronizations. If both the geometry and the mesh are imported, only the CAD geometry and the mesh displayer are made visible. Surface elements, solid elements, and nodes are imported but visually hidden. If only the mesh is imported, solid elements are hidden to avoid clashing with the surface elements. If the mesh does not contain any surface elements, the solid elements are displayed. Nodes are always hidden by default.

The mesh displayer that is created and managed by the TD Direct importer can also be selected and edited. The form shown in Figure 2-9 will be displayed. The label is the text that is attached to the right side of the mesh displayer and may be changed if desired. However, the most likely feature to be used is to set the display preferences.

The mesh displayer serves as a simplified display for all of its nodes and elements. The individual nodes and elements may be hidden and the mesh represented graphically by the mesh displayer. This greatly improves graphics rendering speed. The fastest and visually simplest display is Wireframe Outline. This will suppress all interior element lines and just display a general outline. Use this option for maximum graphics performance when it is not desired to view the individual elemental breakdown.

Two sets of display options are maintained. One for Thermal Desktop model building mode, and one for Thermal Desktop post processing mode. In post processing mode, the mesh is displayed as a shaded representation with hidden interior faces. The mesh displayer displays post processed data for the elements that it manages, allowing the individual elements to remain hidden for better graphics performance.

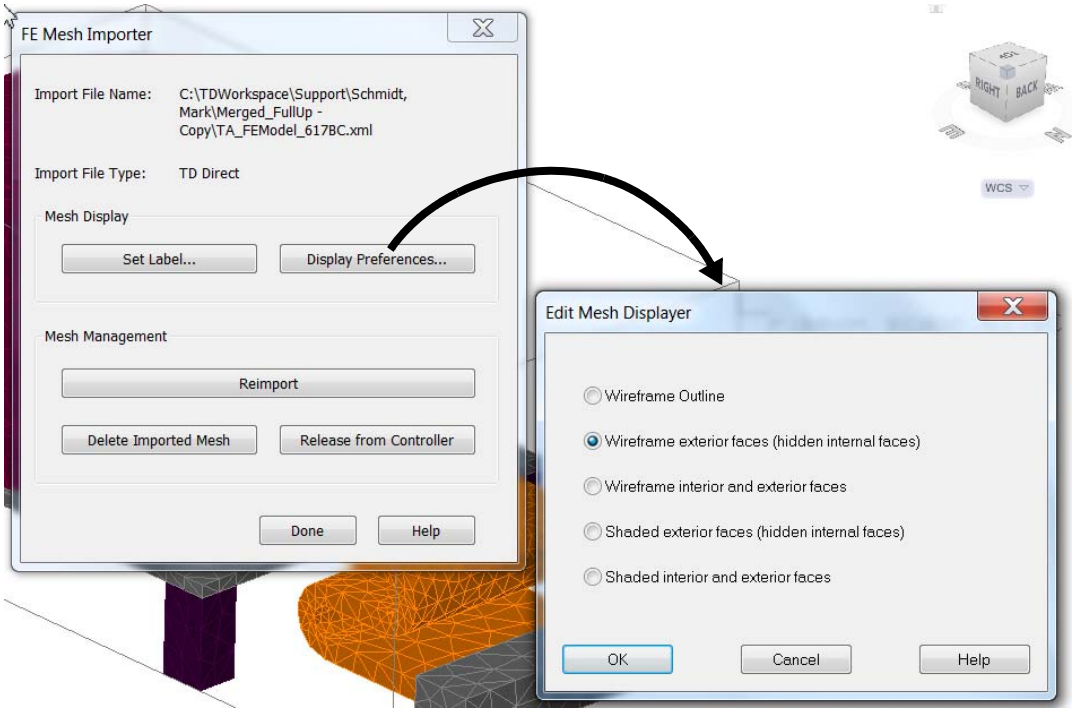


Figure 2-9 Mesh Displayer managed by TD Direct importer may be edited for visual appearance in both model building and post processing display modes

The Mesh Management features will rarely be used, but are provided to mirror the behavior of mesh displayers used elsewhere in Thermal Desktop. The Reimport feature is currently reserved for future expansion. The mesh may be released from the controller and the net result will be a set of independent nodes and elements. This may be useful for one-time modifications by hand, or to export to an older version of Thermal Desktop. More information on mesh displayers can be found in the Advanced Modeling Guide, which can be found under the main Windows Start menu: Start->Thermal Desktop->Users Manual - Meshing.

Geometry imported from TD Direct will be placed on a special layer. Each TD Direct importer creates specific layers for the items that it manages. Geometry is placed on a layer named TDFEM_PRT_<*tddi_handle*>, where <*tddi_handle*> is the AutoCAD handle assigned to the TD Direct importer. In the example in Figure 2-8, the geometry will be placed on a layer named TDFEM_PRT_617BC. This layer may be turned off and on to hide and show the imported geometry. As a shortcut, all imported geometry may turned off and on simultaneously using TD Mesher->Mesher Part Off, and TD Mesher->Mesher Part On.

Layers are also created for the mesh displayer, surface elements, and solid elements. Nodes are placed on the layers corresponding to their parent element. The mesh displayer is placed on a layer named TDFEM_MC_<*md_handle*>, where <*md_handle*> is the handle of the mesh displayer (MC stands for mesh controller, a type of mesh displayer). The mesh displayer handle will differ from the TD Direct importer handle and can be seen attached to the right side of the mesh displayer graphical representation.

By default, surface elements will be placed on a layer named TDFEM_2D_<*md_handle*>, and solid elements will be placed on a layer named TDFEM_3D_<*md_handle*>. Commands similar to turning the part layers on and off are also available for mesh displayers, surface elements, and solid elements. The tool bar for controlling layer visibility for layers created by TD Direct importers is shown in Figure 2-10. These are global commands that turn on or off layers for all TD Direct importer entities simultaneously.

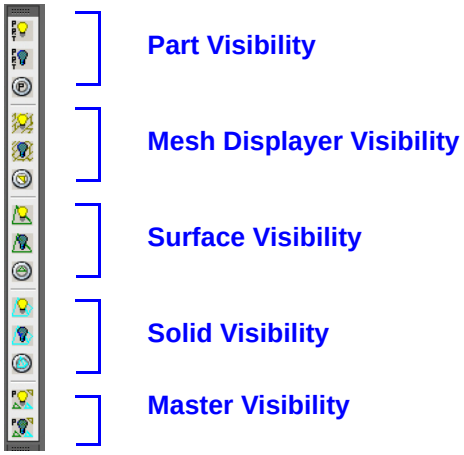


Figure 2-10 The Mesher Layer Visibility tool bar controls the visibility of TD Direct importer items

Layers for mesh entities can also be specified using any desired name in TD Direct. If no layer is specified in TD Direct, the above default layers will be used. However, if geometry is placed on a layer in TD Direct, mesh entities that are created will be placed on the same layer name with the same layer color in Thermal Desktop. The underlying SpaceClaim tool in TD Direct provides a layer feature that is similar to the layer feature in AutoCAD.

2.3 Editing a TD Direct Importer

A TD Direct importer can be modified and synchronized by selecting the bounding box and Thermal > Edit. The importer not only controls importing geometry and creating a mesh, but may also send parametric dimensions to TD Direct to modify the geometry before importing and meshing. The importer can apply editing actions after import of the mesh to domains defined in TD Direct, for example, to change all nodes on an edge to be boundary nodes. The following sections discuss the features of the TD Direct importer in detail.

2.3.1 Options Tab

The two choices for the Options Tab, shown in Figure 2-7, are Import CAD Geometry and Generate Finite Element Mesh. The two options can be chosen at the same time, individually, or not at all. The transfer of geometry from TD Direct to Thermal Desktop

occurs behind the scenes, but uses ACIS as the transfer format. If there is an error transferring geometry using ACIS V6, try using ACIS V7. Geometry imported successfully is identical with using either format option, and has no effect on mesh generation.

If Import CAD Geometry is selected, the engineering geometry in the TD Direct document will be imported into Thermal Desktop upon synchronization. The imported geometry can be used for scaffolding to assist FD model building or to verify proper alignment and positioning for the mesh that will be generated in subsequent synchronizations. If the geometry in TD Direct changes, the updated geometry can be brought into Thermal Desktop by synchronizing all links (Section 2.4), or by editing an importer and selecting the Synchronize button. Importing geometry automatically converts the geometry dimensions from TD Direct length units to the current Thermal Desktop units.

Checking the Generate Finite Element Mesh checkbox will generate a new finite element mesh in Thermal Desktop based on the mesh controls (Section 4.3) and thermal tags (Section 4.4) defined in the TD Direct session. If a mesh currently exists, it will be replaced by the new mesh upon synchronization. Any existing connections to other portions of the thermal model that were made using domain tag sets will be preserved. Connections made without using domain tag sets will be lost.

2.3.2 Dimension Overrides

SpaceClaim allows the parameterization of geometry using *Driving Dimensions*, and TD Direct enables access to this feature from within Thermal Desktop.

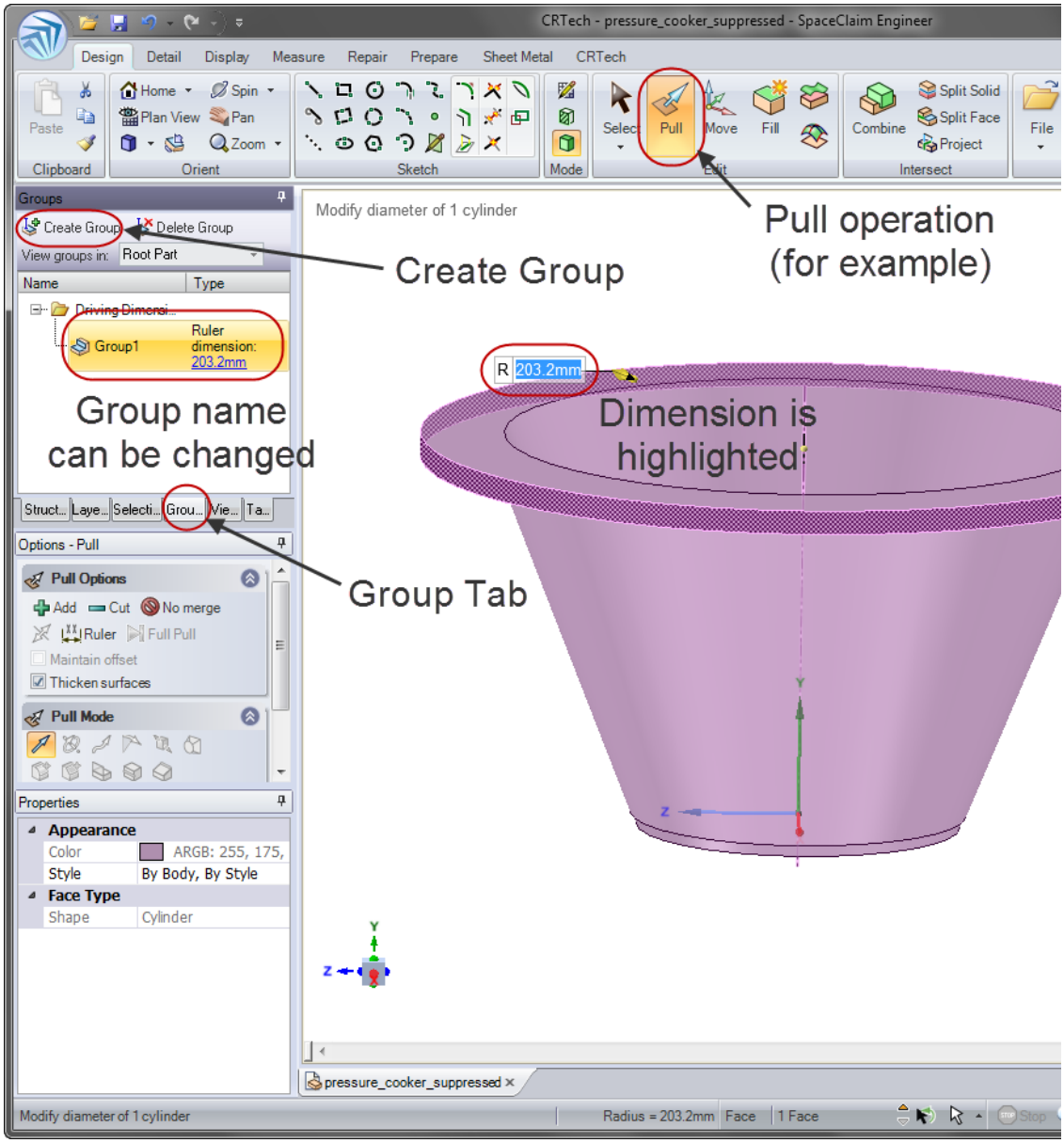


Figure 2-11 Creating driving dimensions in SpaceClaim

Driving dimensions are created in SpaceClaim by creating a group on the Group tab while a dimension is highlighted within an operation, such as Pull (Figure 2-11). Once the SpaceClaim group is created the name can be changed and the dimension edited by selecting the value and typing a new value. Using the TD Direct importer in Thermal Desktop, any driving dimension in the TD Direct session can be changed from within the Thermal Desktop session by using the Dimension Overrides tab on the importer form (Figure 2-12).

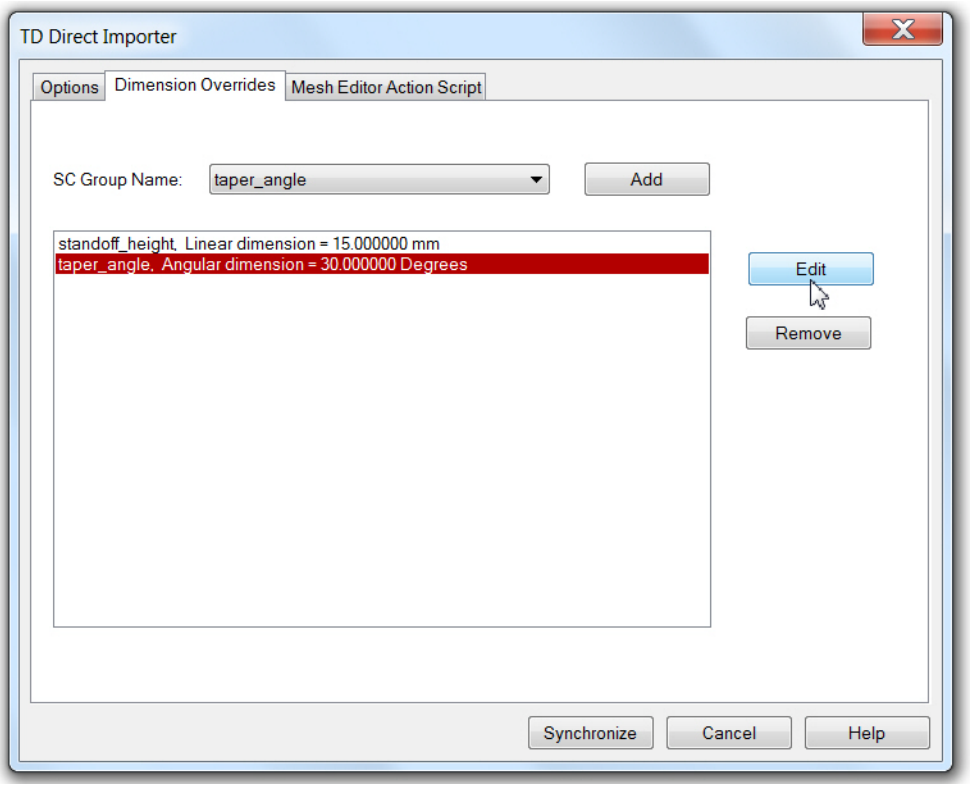


Figure 2-12 Dimension override tab in the TD Direct Importer editing form

To control the driving dimension from Thermal Desktop, select the driving dimension group name from the SC Group Name pull-down menu and select Add. The Edit Driving Dimension form will open with the current value of the driving dimension in the field in Thermal Desktop units. Changing the value and selecting OK will update the SpaceClaim driving dimension on the next synchronize operation.

The updated geometry will then be reimported into Thermal Desktop, with a correspondingly updated mesh if the mesh generation is requested on the Options tab. The value in the Edit Driving Dimension form can be provided by a Thermal Desktop symbol by double-clicking on the field to access the Expression Editor. See the Thermal Desktop User's Manual for more information on symbols and the Expression Editor.

Important: It is the user's responsibility to ensure changes to the driving dimensions are reasonable and valid.

2.3.3 Mesh Editor Action Script

This form allows the user to define post-import editing actions that operate on domain tags defined in TD Direct. A typical application would be to change the nodes in a specified domain from diffusion to boundary nodes.

The edit options are Nodes, Nodes No Id, Surfaces, and Solids. First, the domain is selected from the Domain pull-down menu. Then the editor type is selected from the Editor Type pull-down menu. Depending on the editor type, a particular tag set generated by the

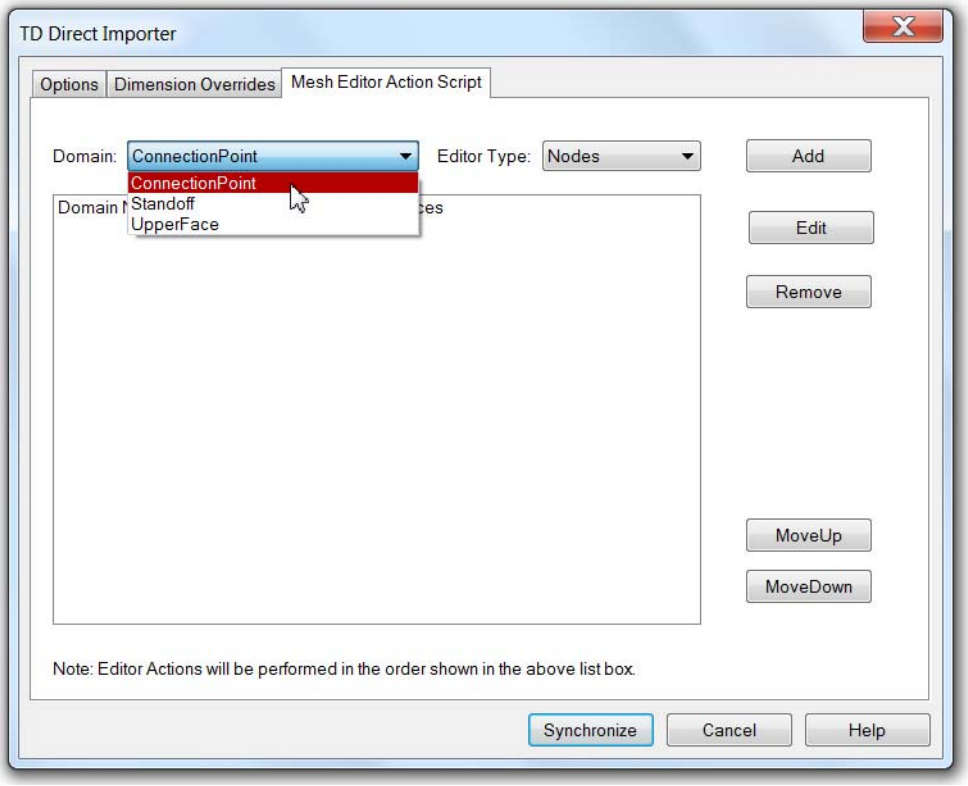


Figure 2-13 Mesh Editor Action Scripts for TD Direct Importer

selected domain will be edited. In the above example, the surfaces in the domain tag set "external_convection_surfaces" will be edited for the second selection in the list. The domain name and the editor type determine the tag sets used in the editor action.

After selecting the Add button, the appropriate edit form is opened. Functions that may be accomplished with editor actions are (but not limited to):

- setting initial temperatures
- changing nodes to boundary or other node types
- defining insulations made of material stacks
- setting conductivity and density multipliers
- adding comments
- defining free-molecular conduction

The editor actions are performed in the order listed when a synchronization is requested. The order can be changed by selecting the desired action and clicking the Move Up or Move Down button.

Editor actions may be edited or removed by selecting the appropriate button.

Note: For editor actions, all values for all objects in the domain tag set will be changed to the values shown in the edit form. Be sure the editor action is filled in with all desired values. An exception is the Nodes No Id editor, which will leave the node Id's unchanged, however, values in all other fields will be applied to the nodes in the domain tag set.

Note: Expansions to the editing scripts are planned for future releases to allow finer granularity in setting parameters.

Domains, Tags, and Domain Tag Sets are described in more detail in Section 4.4.4 "Domains".

2.4 Synchronization

When all desired options have been set for the TD Direct importer, use the Synchronize button to execute those options. Upon synchronization the following steps are taken in order:

- The referenced TD Direct document is opened, if it is not already open. SpaceClaim/TD Direct will be launched if necessary.
- Driving dimensions from Thermal Desktop are sent to TD Direct and the geometry is updated.
- Material and optical property names, submodels, and radiation analysis groups used in the Thermal Desktop model are sent to TD Direct to be made available within the TD Direct pull-down editing forms.
- The updated geometry is imported from TD Direct and/or a finite element mesh is generated and imported based on the updated geometry.
- The mesh editor action scripts are performed on the specified domains.

Synchronization can be performed on a single importer by selecting and editing (Thermal > Edit) the importer bounding box. All TD Direct importers in the Thermal Desktop drawing can be synchronized simultaneously by selecting Thermal > TD Direct > Synchronize Links.

Note: The Create New Importer ***and*** Synchronize All Importers ***buttons on the tool bar are directly next to each other; use care when selecting the desired operation.***

3 Creating, Importing, and Preparing Geometry

SpaceClaim Engineer represents a revolution in CAD technology. It is the first CAD tool designed specifically for the engineer with the intent of producing geometry for engineering analysis. SpaceClaim's goal is simple: to provide engineers with advanced 3D modeling capabilities without having to become CAD experts. It has had critical acclaim, including being named NASA Tech Briefs' Readers' Choice Product of the Year for 2012.

SpaceClaim Engineer (or simply, SpaceClaim) allows geometry to be created natively or imported from almost all CAD formats as well as vendor neutral formats such as STEP and IGES. Imported geometry is prepared for analysis using powerful defeaturing, simplification, and healing tools.

It is a *direct modeler*, in that it allows geometry to be modified without the need for a history tree. Faces, edges and features are directly manipulated by interactive pulling, filling, and move operations. This significantly flattens the learning curve and empowers the analyst to explore solutions rapidly.

In a typical engineering organization, there is a long slow loop between design iterations. The design model for manufacturing is created by the designer in the company's preferred CAD system, which is passed to the engineer, who builds a separate representation appropriate for analysis, and then feeds the results back to the design team. If a change is needed, the process starts over again, essentially from scratch. It is even worse in a multi-disciplinary environment, such as one requiring the analysis of thermal structural deformations or optical systems. Each discipline's engineering models rapidly diverge, making it more difficult to map engineering information, such as temperatures on to a structural model, as successive design iterations occur.

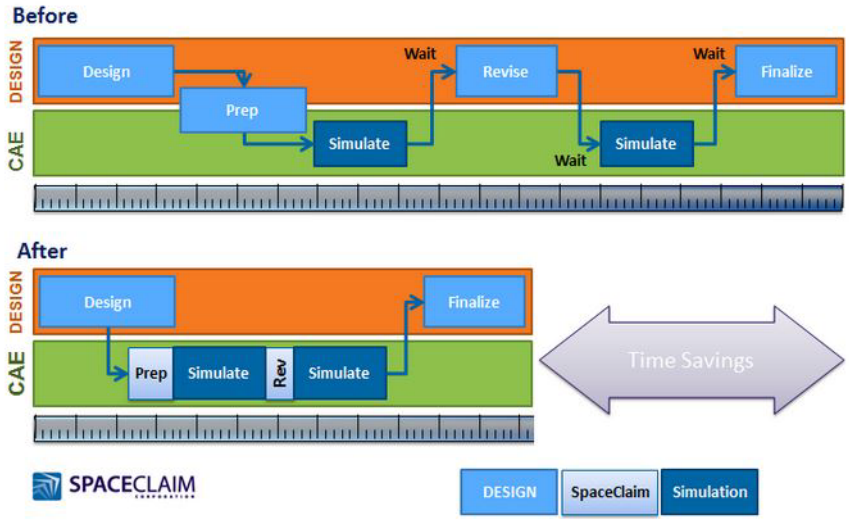


Figure 3-14 SpaceClaim improves cycle time with efficient preparation tools and the ability to perform trades without needing assistance from the CAD designer. (Image provided courtesy of SpaceClaim Corporation)

Each time the design changes, either to explore “what if” scenarios or to correct a defect, a lengthy process ensues. SpaceClaim provides capabilities for rapid conversion of design geometry into a representation appropriate for analysis, keeping the thermal model concurrent with the latest design and subsequently other engineering discipline’s analysis models.

SpaceClaim allows easy modification of geometry for performing trade studies. It allows native construction of geometry from scratch, allowing the thermal engineer to proceed in the early concept phases of a project, or to provide geometry not present in the detailed design model, such as test chamber enclosures, lamp arrays, or other test fixtures.

The following sections provide an overview of the capabilities of SpaceClaim for creating, importing, and preparing geometry. The user is highly encouraged to become familiar with the basic operations of SpaceClaim before attempting an analysis project. The time spent up front covering the basics is well worth the effort. SpaceClaim has a very intuitive interface, and many features can be learned by discovery, however, it is highly recommended to take advantage of the excellent training resources provide by SpaceClaim. The following resources should be utilized:

Online Help. By clicking the question mark (?) in the tab bar or pressing the F1 key, the user has access to help within SpaceClaim. The online help provides step-by-step instructions, animations, and examples. Within the online help are also a printable quick-reference card for keyboard shortcuts and another for mouse and touch gestures (movements with a mouse or on a touchpad or touchscreen to issues commands).

SpaceClaim Website (www.spaceclaim.com). Under the support section of the SpaceClaim website are tutorials and downloads that can be used to learn the features provided by SpaceClaim. A good place to start is the Basic Tutorials, http://www.spaceclaim.com/en/Support/Tutorials/Essentials/SpaceClaim_Basics_Tutorials.aspx.

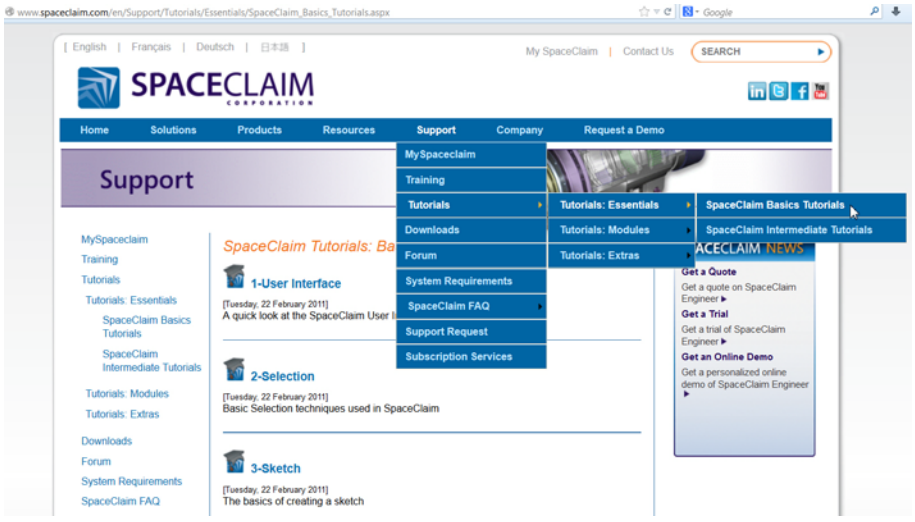


Figure 3-15 Comprehensive video tutorials can be found at the SpaceClaim website

CRTech Website (www.crtech.com). An overview of SpaceClaim Engineer for geometry preparation, and a follow-on for an overview of meshing are available at http://www.crtech.com/spaceclaim.html.

3.1 Creating Geometry

Thermal Desktop contains a built-in suite of “primitives” that may be used to represent 2D conic surfaces trimmed on regular parametric boundaries (discs, paraboloids, cones, etc.) as well as a collection of regular 3D solid primitives (cylinders, spheres, etc.). When those simple shapes are insufficient to capture the behavior of a system, CAE tools such as SpaceClaim in conjunction with meshing should be employed. (It should be noted that TD Direct allows the analyst to mix and match as desired the use of finite difference primitives as well as finite element meshes generated from arbitrary geometry.)

With the advent of SpaceClaim, many engineers are creating analysis-ready design geometry from scratch themselves, rather than simplifying the complex detailed design geometry intended as a complete specification for manufacture. Often a combination of importing/simplifying detailed design geometry *and* creating geometry natively in SpaceClaim is the best approach for generating an engineering appropriate representation. This section provides a brief overview of the geometry creation capabilities in SpaceClaim and is intended as an introduction. Please consult the SpaceClaim documentation and tutorials for a complete reference on geometry creation functionality.

Geometry created by the thermal analyst in SpaceClaim is referred to as “engineering” geometry, to denote that it is not intended for manufacture (detailed design geometry) nor directly used for generating thermal results (analysis geometry). Engineering geometry is geometry generated by SpaceClaim that is used to create a thermal model by finite element meshing or by extracting key information for finite difference primitive creation. Thermal Desktop analysis geometry is created by TD Direct from the engineering geometry created in SpaceClaim (see Section 4 "Meshing with TD Direct").

A simple example of a tank is presented here to highlight some of the many ways to create geometry. One way to start is by sketching curves on a sketch plane. The sketch plane may be located anywhere in space, and existing faces and edges can be used to place the sketch plane. A sketch plane can be moved and rotated at any time.

Sketched curves are then pulled into surface or solid shapes. The pull can be normal to the sketch plane, along a path or revolved around an axis. Blends can be performed between two arbitrarily shaped and positioned curves to create a surface, or between two surfaces to create a solid.

In this example, we start with a blank drawing in which the sketch plane is located at the origin in the XY plane. The sketch mode is enabled by selecting the Sketch Mode button. In sketch mode, the tools on the Sketch tab may be used to create and modify various types of curves. Figure 3-16 shows a circle created by dragging the mouse. The radius is displayed in a text box. The mouse may be used to click at a location to set the radius, or a number entered at any time. Curves can be edited and values changed at any time.

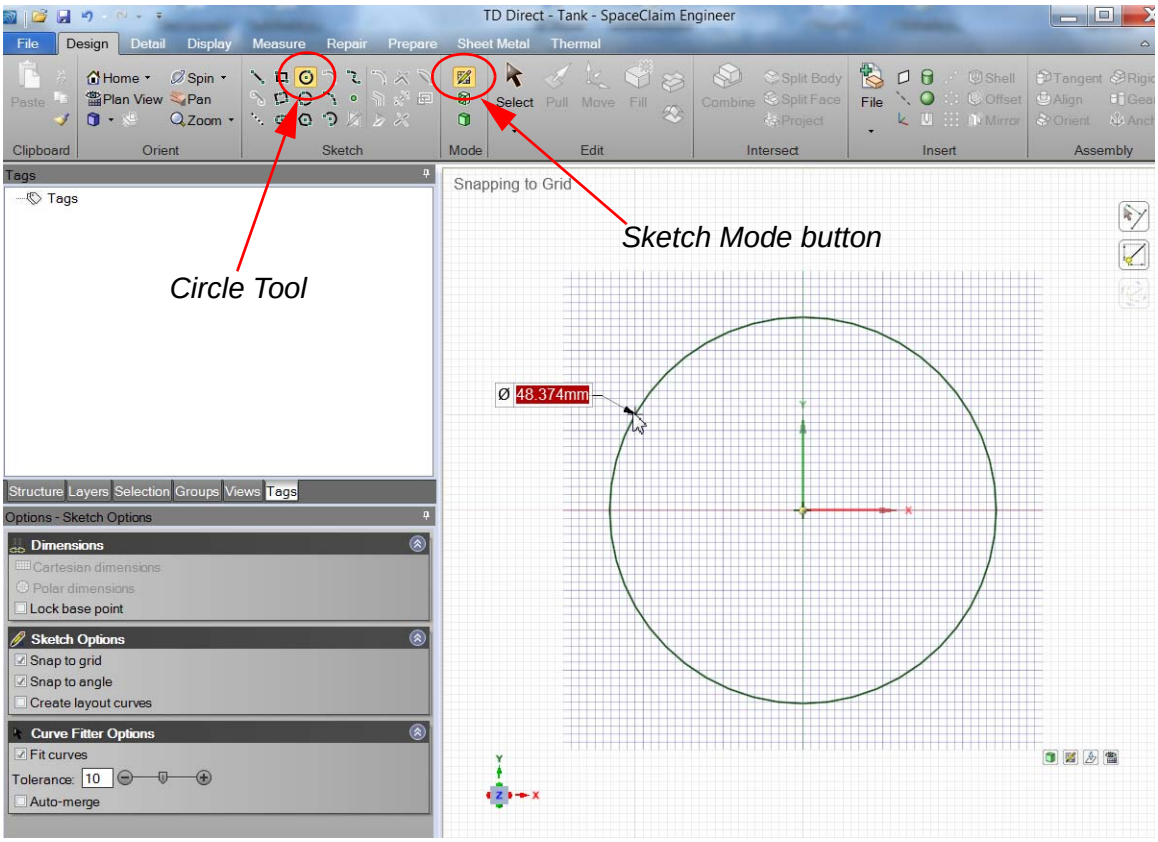


Figure 3-16 Sketch mode is used to create curves on the sketch plane

After the desired curves are created, select the 3D Mode button to work in 3D space. Closed curves are automatically converted to surfaces. Rotate the view as desired and use the Pull tool to create surfaces or solids from curves. Select the Pull tool, and then click on the face to pull it into a solid cylinder. Many options exist for the Pull tool, it is one of the main tools used for creating geometry. The solid cylinder is shown in Figure 3-17.

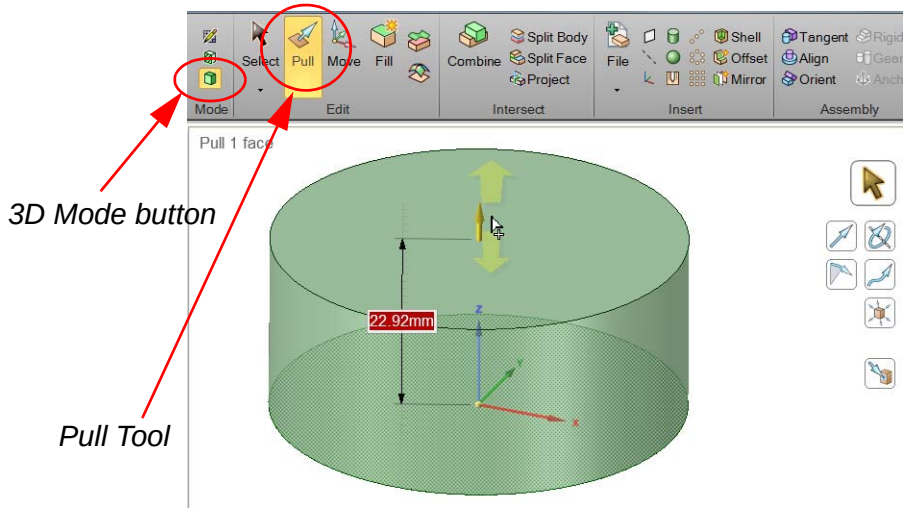


Figure 3-17 The Pull tool is used to create solids and surfaces from sketched curves

Figure 3-18 shows the model after a hemisphere was placed on the top of the cylinder, and the part mirrored about the XY plane. An exit pipe for the tank was created by placing the sketch plane in the XZ plane, drawing a circle then pulling in the -Y direction. The inlet pipe is created by placing the sketch plane at the base of the hemisphere, and then drawing a circle tangent to the radius of the tank. SpaceClaim recognizes tangent, perpendicular, parallel, and other conditions and automatically builds in the constraints. If the diameter of the circle is changed, it will remain tangent to the outer radius of the tank. The pipe is created by switching to 3D mode and pulling the circle into the tube.

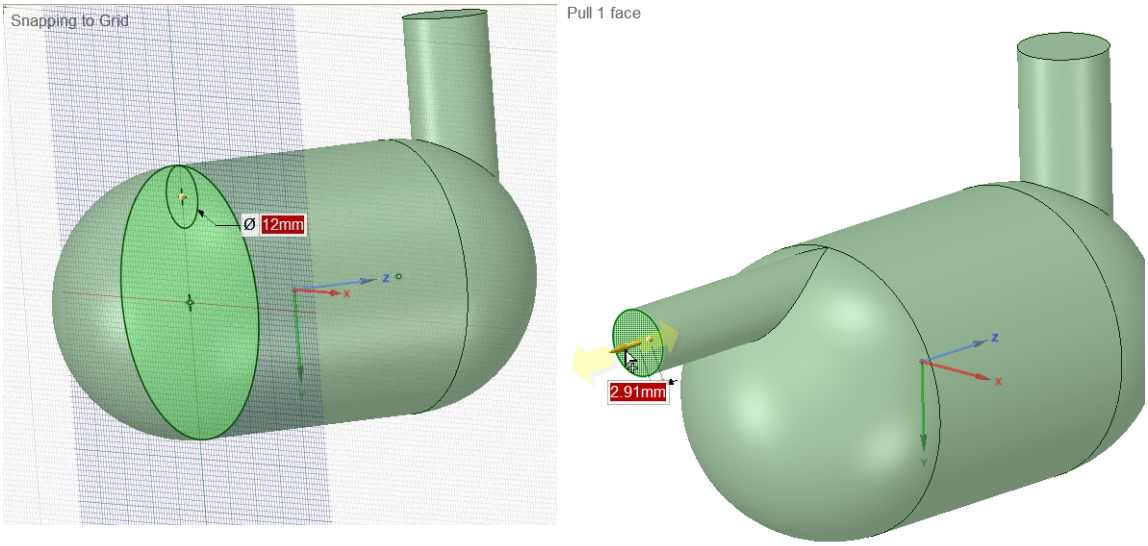


Figure 3-18 The sketch plane is placed at the base of the hemisphere, a tangent circle is drawn, then pulled to create the inlet pipe

This example shows a solid part being created. Since we are creating a tank, we need to have a surface model. It is entirely possible to work with surfaces from the start, but it is often easier to create a solid model and let SpaceClaim automatically combine the pieces into a single solid. A solid object can then be converted into a shell of surfaces by right-clicking and using the Detach command.

There are three main tools for creating and modifying geometry. In addition to the capabilities just described, the Pull tool can also create rounds, chamfers, and fillets by selecting an edge (double click to select an edge-chain). The Move tool repositions and makes copies of objects. The Combine tool is used for both merging and cutting objects. It supplies all the possible boolean operations that can be done between objects.

On the left side of Figure 3-19 is the Structure Panel. It shows a single *component*, “Tank”, that consists of five surfaces. The structure panel permits the creation and organization of any number of components. Components can contain other components to make a hierarchical assembly. Components can be created in the current document, or included externally from other documents. Using external components is a good way of managing a large complex assembly.

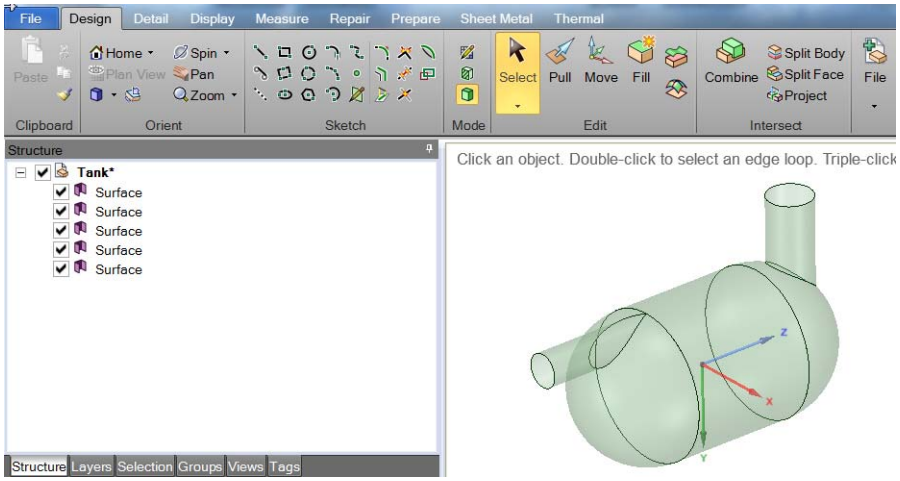


Figure 3-19 A solid part is easily converted into surfaces using the Detach command

SpaceClaim contains many, many, powerful features for creating geometry. Assemblies may be created with constraints between components. Tangent, coaxial, and gear mating conditions can be applied to facilitate the creation of an assembly or simulate the motion of a mechanism. A wide range of display options are available to add color and texture for clarity or realism. Items may be grouped on layers, which are transferred to Thermal Desktop when a TD Direct document is synchronized with a Thermal Desktop model. Dimensions may be created that can be directly modified, and even driven by Thermal Desktop.

All of these features are documented in the SpaceClaim Engineer on-line help and web based tutorials. Please take advantage of these resources to become familiar with the many features of SpaceClaim before beginning your thermal analysis project.

3.2 Importing Geometry

Geometry is imported into SpaceClaim by using the File command on the Insert tab. Data from all major CAD systems and vendor neutral formats can be inserted into a TD Direct/SpaceClaim document. After a file type is chosen on the drop down menu, options specific to that file type will appear on the form. It is recommended to choose all of the improvement and checking options that are presented.

A file will be inserted into the *active component*, which is indicated by a bold type font in the Structure Panel. When a document is first created, the active component is the top level item in the tree and is named the same as the document. Components can be created under a component by selecting the component in the Structure Panel, right-clicking and choosing New Component. A component is made the active component by right-clicking the desired component and selecting Activate Component.

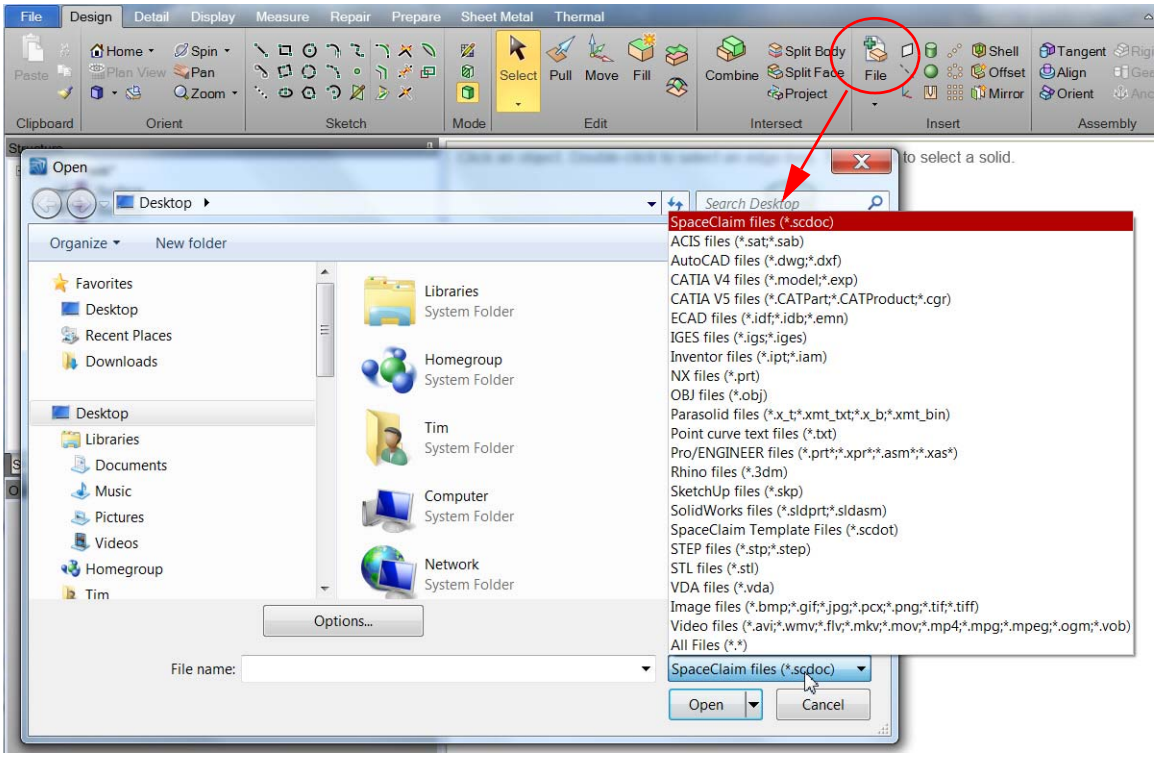


Figure 3-20 The File command on the Insert tab is used to import files into a document from all major CAD systems and formats

The active component will receive the contents of an import operation and any native creation operations done using SpaceClaim. If more than one external CAD drawing is being imported into a single document, create new components before importing to keep the assembly structure more manageable.

Often only a subset of a full system level model needs to be analyzed. If only the full system level design model is available, import the full-up model and then extract just the needed items. After importing a complex assembly, any component may be placed in its own document. To make a component *external*, right-click on the component and select Source > Convert to External. A separate document with the name of the component will be created. Use this document to synchronize with Thermal Desktop.

Each CAD system has many options when exporting data. In general, the best results are when the native CAD system file is imported. However, it may be the policy to export only in STEP or IGES format. These formats allow the entire assembly to exported as a single part. If possible, request from the designer that assembly information be retained when exporting to STEP or IGES.

Please consult the SpaceClaim on-line help and tutorials for more information on managing components and import options.

3.3 Preparation

Very rarely can an imported design model be used directly for analysis. The amount of detail usually present leads to intractable analysis models. In addition, imported geometry often contains defects that prevent a mesh from being generated. The requirements on geometry for meshing are more strict than required for a purely visual representation.

Geometry for meshing must be topologically precise. That is, a solid must be bound by surfaces that all share common edges. The edges on a surface must share common vertices. Geometry must be “water tight”, in that no gaps exist between surfaces that form a solid. This requirement is not strictly necessary for an acceptable visual representation, but is required for meshing.

Even if geometry is topologically sound, it can still contain unacceptable items that prevent successful meshing. Very small edges, cracks, and sliver faces may exist in the model. Parts may slightly interfere or have gaps where they should mate perfectly. Fortunately for the engineer, SpaceClaim provides the features necessary to repair, defeature, and simplify geometry in preparation for meshing.

3.3.1 Repair

Repair consists of correcting topological defects, such as filling in missing surfaces, and combining edges of adjacent surfaces to form a valid solid. It may also consists of removing interfering regions, or extending geometry such that it mates properly. Consider the model shown in Figure 3-21, taken from the SpaceClaim Repair tutorial.

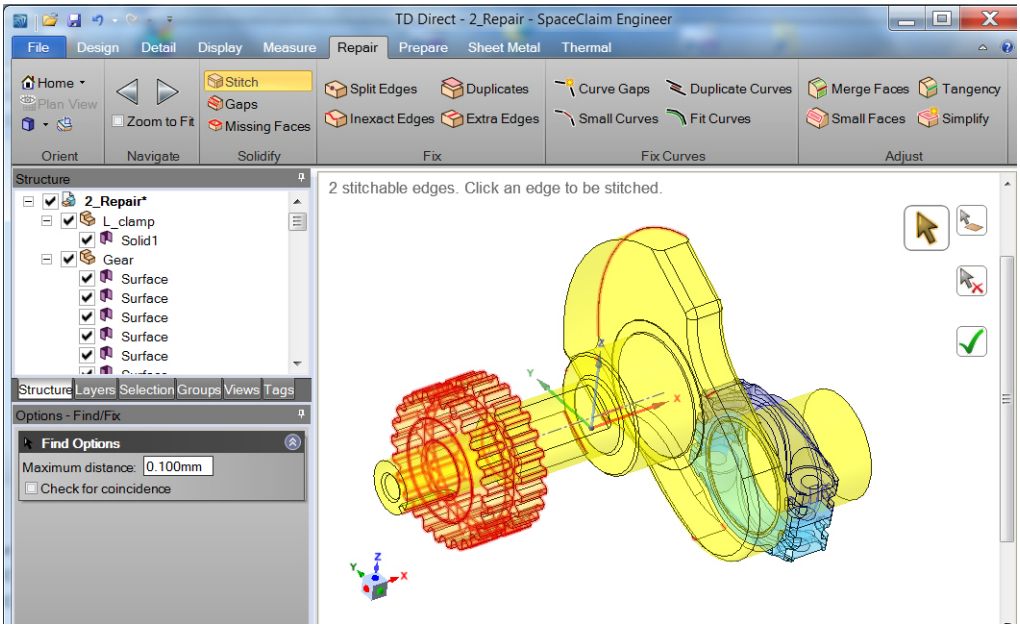


Figure 3-21 Repair functions correct defects in geometry in preparation for meshing

Notice that the structure tree indicates the model is composed of surfaces, when it should be composed of solid parts. This is an indication that the topology is incorrect. The Stitch tool in the Solidify group attempts to combine edges of surfaces in order to form a water-tight solid. Figure 3-22 shows the results after stitching. It can be seen that the gear now appears solid, but the rest of the assembly still contains some problem areas. The Gaps tool searches the geometry for regions of mismatched edges. Gaps here refer to gaps in the surface topology, not gaps as a space between solid parts. The counterweight of the crank-shaft contains a defect, which is shown enlarged in the right viewport.

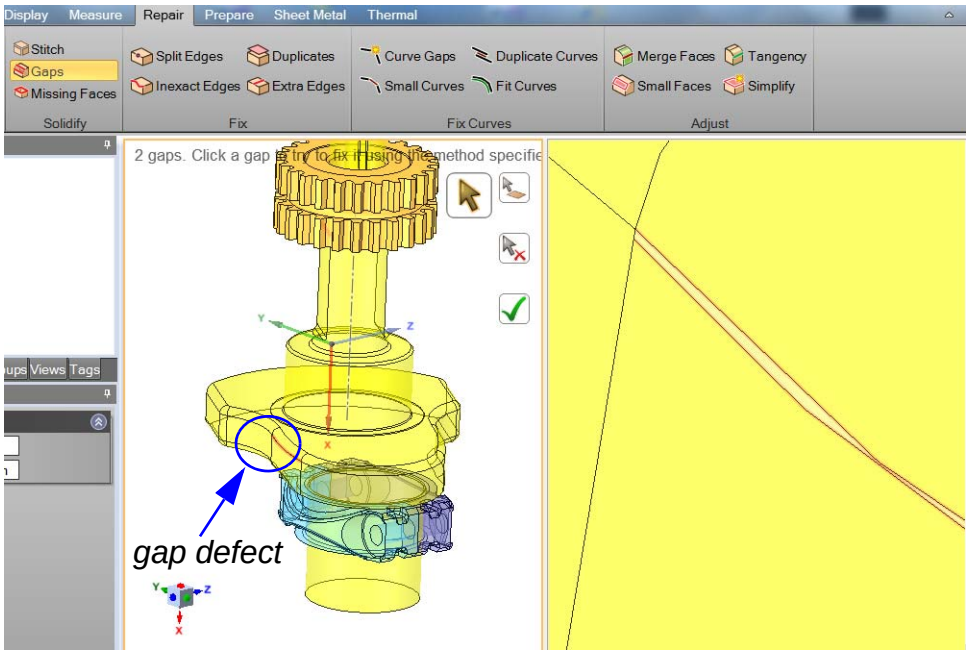


Figure 3-22 The Gaps tool finds and repairs defects in surface topology

Selecting the green check mark icon in the graphics area will complete the gaps repair operation. This model, however, still has additional defects. The Missing Faces tool discovers many missing faces on the bearing caps, as shown in Figure 3-23.

The Missing Faces tool will attempt to create surfaces on an open collection of surfaces in order to create a solid body. Do not use the Missing Faces tool if you intend to have a surface model.

After fixing solid defects, perform all the checks in the Fix and Fix Curves groups. This will combine edges that have an intermediate and unnecessary vertex, refine edges that are inexact, and remove overlapping duplicated items. It may be necessary to cycle through the fixes a number of times to completely remove all defects.

The Simplify tool in the Adjust group will examine the geometry to see if spline surfaces can be replaced by exact mathematical representations, such as planes, cylinders, or spheres. After examining and repairing geometry using the tools on the Repair tab, use the tools on the Measure tab for further examination of the geometry.

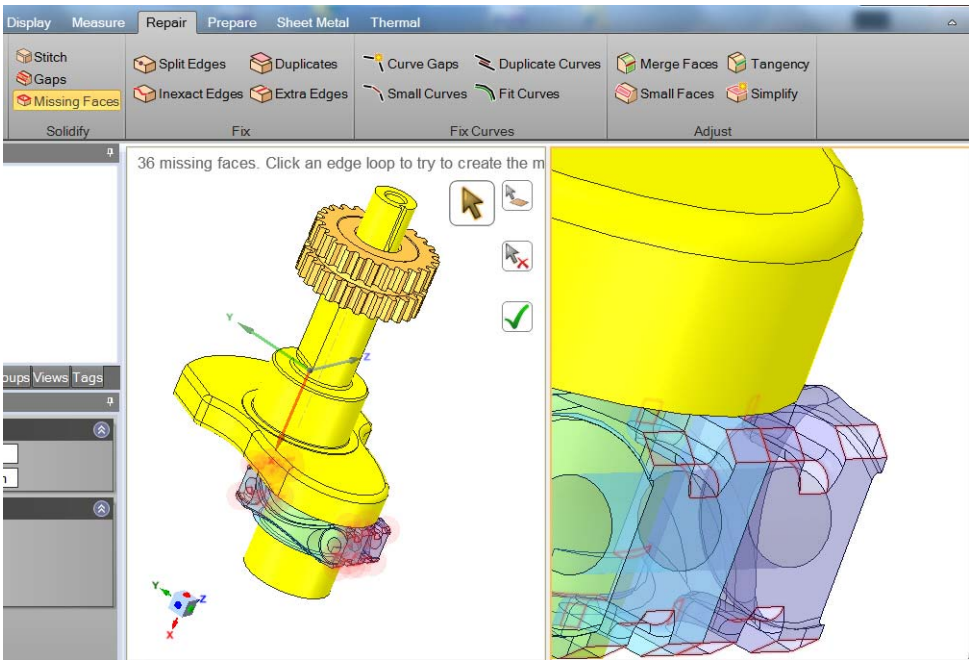


Figure 3-23 The Missing Faces tool will patch open surface bodies into solid bodies

To perform a thorough check of the entire model, including tests for non-manifold conditions, select the top level component from the structure tree, and then select Check Geometry in the Inspect group. The Measure tab, shown in Figure 3-23, contains a few other useful functions.

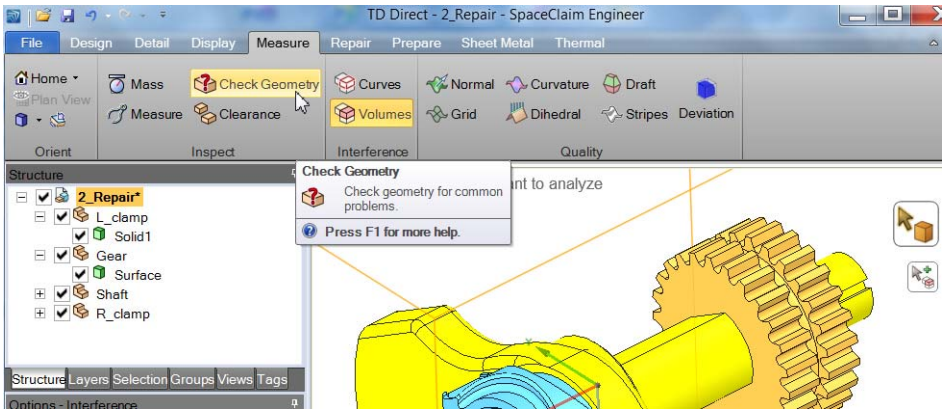


Figure 3-24 The Measure tab contains tools for checking for internal consistency, clearances, and interferences

Often a failure to mesh is caused by intersecting solids. It is also possible that what is desired to be a contiguous part is actually disconnected. Use the checks in the Interference group to find areas that share the same space. Use the Clearance tool in the Inspect group to find areas that are close in proximity but do not contact. Correct these geometric imperfections using the geometry modification methods outlined in Section 3.1 "Creating Geometry". The Combine tool may be used to remove intersecting regions, and the Pull tool may be used to eliminate unwanted clearances.

3.3.2 Defeaturing

Once the geometry has passed all repair checks, small unwanted features can now be removed. Defeaturing consists of removing rounds, fillets, chamfers, small holes, notches, pockets and any other small features that have little or no effect on the resulting thermal solution.

The Fill tool and the Selection panel combine to make defeaturing almost trivial. Figure 3-25 shows an electronic box that we desire to remove chamfers, holes and rounds before proceeding to analysis. To remove rounds and fillets, select one that you wish to remove. In the Selection panel, options are present to automatically select other rounds based on size. In this case, the rule “all rounds equal to or smaller” than the radius of the initially selected round is chosen. This searches the model and automatically includes all other items that meet the rule in the selection set. Selecting the Fill tool completes the operation

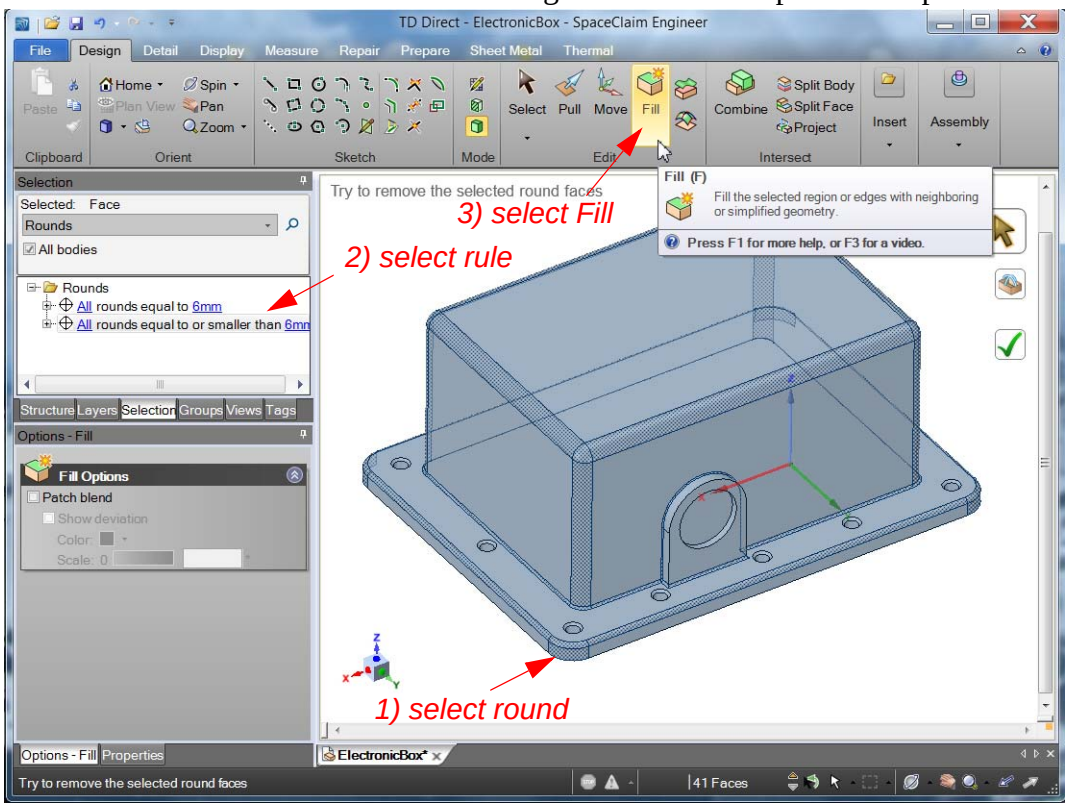


Figure 3-25 The Fill tool combined with the Selection panel makes defeaturing simple

SpaceClaim tools, such as Pull, Move, and Fill remain in effect until another tool is selected. They are not one-time commands, but modes of operation. Additional icons are presented in the graphics area that provide options for the tool that is currently active. This saves time when, for example, many Fill operations are to be performed successively.

Even complex intersecting rounds and fillets can be removed with the Fill tool. Other geometric items besides standard features may be eliminated. Even the connection boss on the side of the box is easily removed with the Fill tool. The bolt holes are all removed in a single step, and in less than a minute the box is ready for analysis.

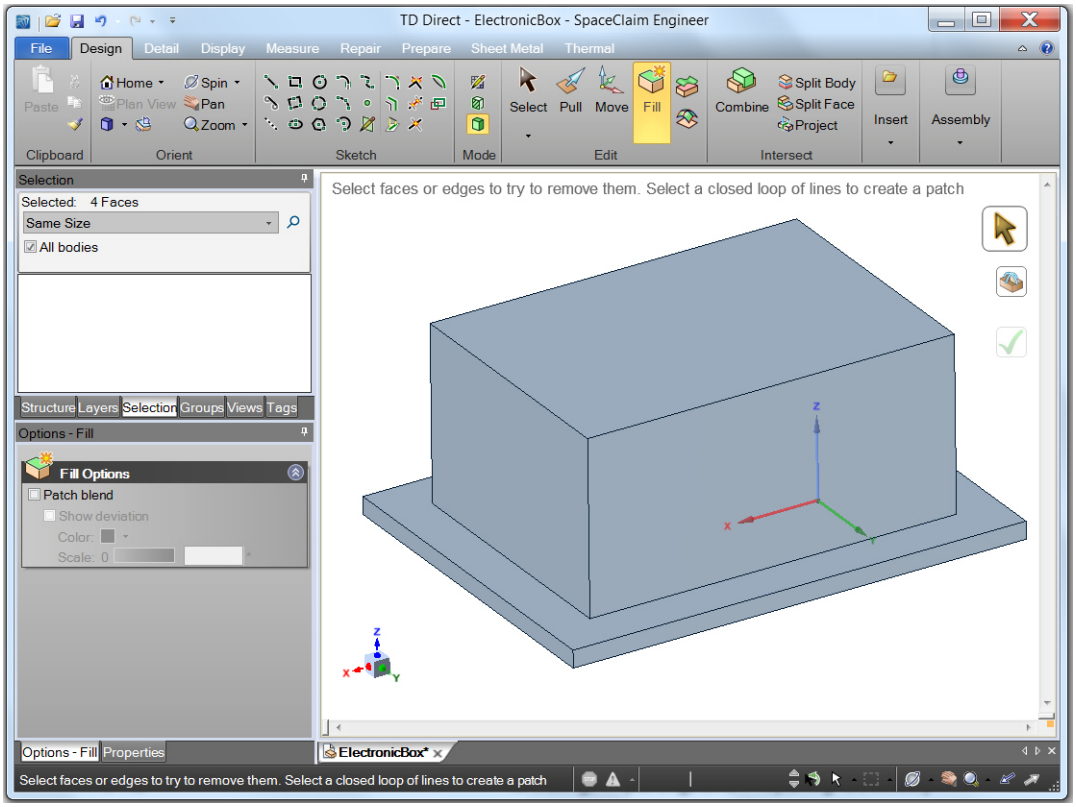


Figure 3-26 The defeatured box is ready for analysis

The Pull tool can also be utilized to defeature complex geometry. The defeaturing tutorial available at the SpaceClaim website shows how to use the Pull and Fill tools in conjunction with cross section mode to quickly turn the complicated splined shaft shown in Figure 3-27 into geometry manageable for analysis.

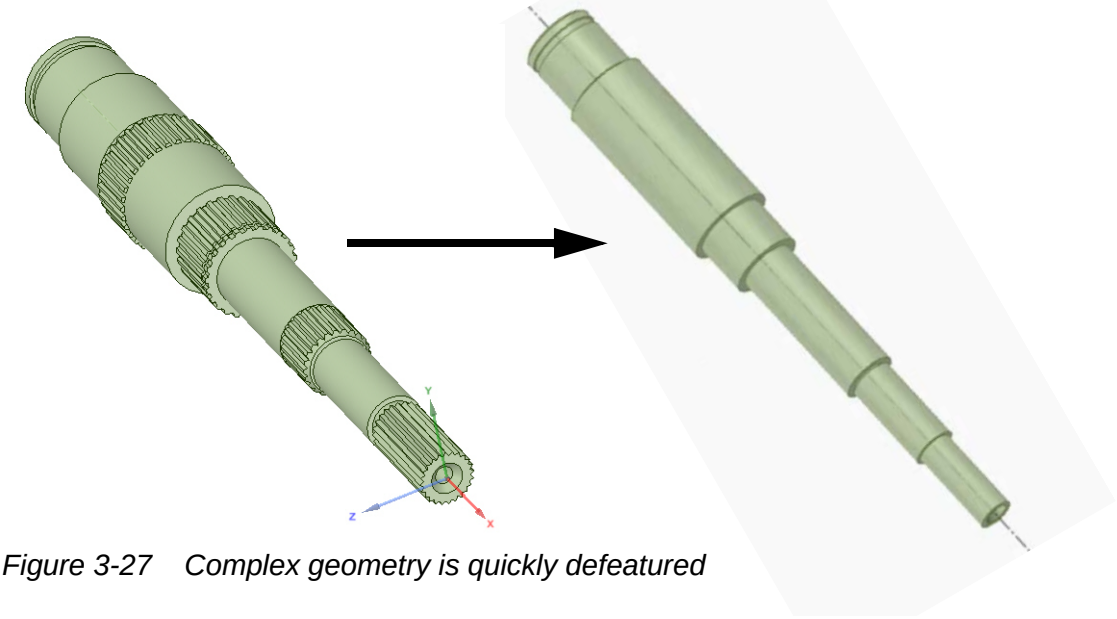


Figure 3-27 Complex geometry is quickly defeatured

Please consult the SpaceClaim on-line help and tutorials, and the CRTech website for more information on defeaturing.

3.3.3 Simplification

We have seen techniques to repair, and defeature parts in preparation for meshing. Often more extensive simplifications are desired beyond fixing defects and eliminating small unimportant features. All of the SpaceClaim tools can be employed to generate an appropriate engineering abstraction of the detailed design.

A common action is to convert thin solids into surfaces. We have already seen in Figure 3-19 how the Detach command can be used to extract surfaces from a solid representation. Another powerful feature of SpaceClaim is the Midsurface tool which can be found in the Define group on the Prepare tab. Lets take a look again at the electronic box cover we defeatured in Section 3.3.2. Suppose that gradients through the thickness of the cover are not important, and in addition a stiffening rib is present in the interior of the box as shown in Figure 3-28.

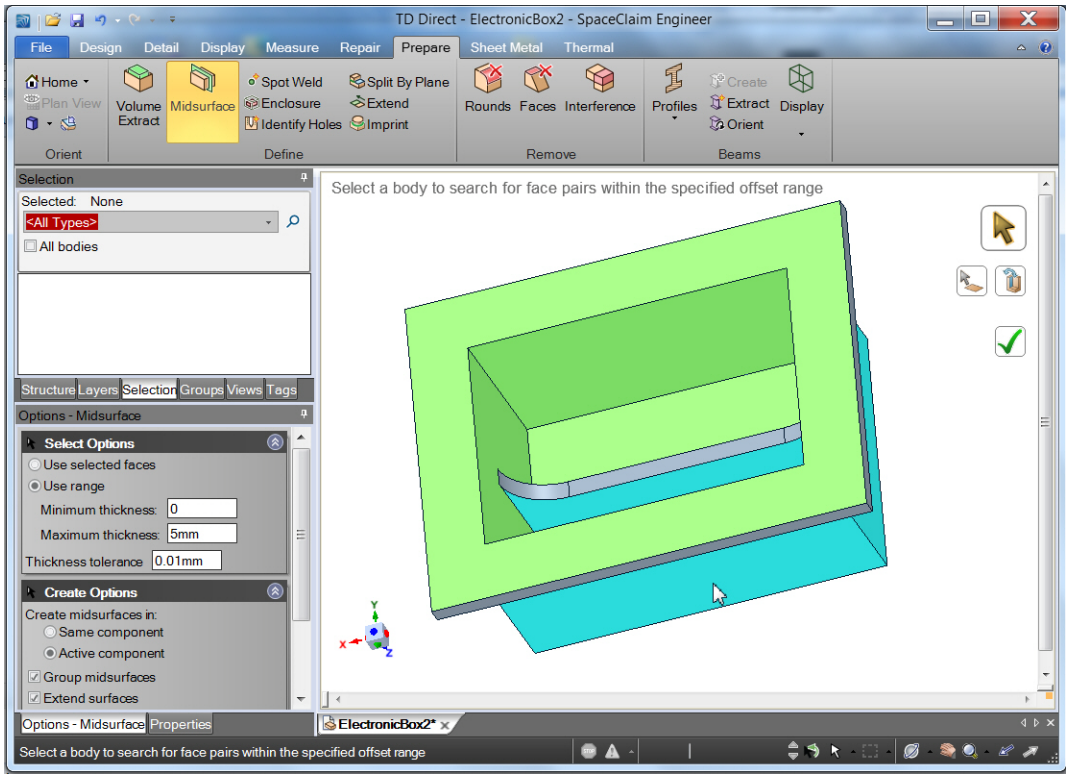


Figure 3-28 The Midsurface tool is used to extract and extend midsurfaces

The Midsurface tool can be used to automatically detect pairs of opposing faces that are good candidates for generating a mid surface. The Use range option is employed in this case, and when a face is selected, all other face pairs within thickness range are automatically selected.

Selecting the green check mark icon will complete the operation. A set of surfaces is generated and automatically extended and trimmed so that they form contiguous geometry. Surfaces could have been detached from this solid, but then many would have to be pulled, trimmed, and moved to form a contiguous part. The Midsurface tool saves a great deal of time by performing this task automatically.

The new set of surfaces can be placed in the same component as the solid, or in the currently active component. The result of the midsurface operation is shown in Figure 3-29.

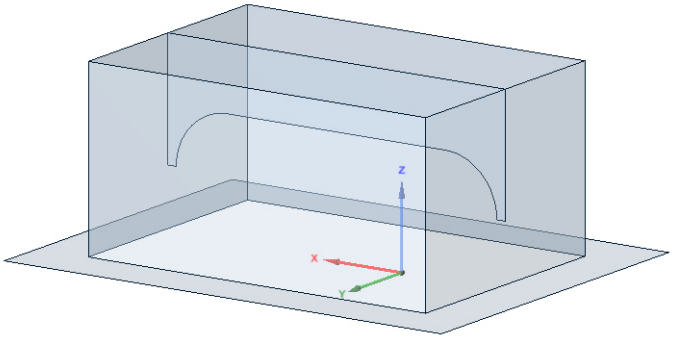


Figure 3-29 Surface representation generated from the Midsurface tool

It is recommended to thoroughly study the documentation on the Midsurface tool before usage. Many options are available, and proper defeaturing and preparation before midsurfacing is essential. The Extend function can be used at any time, and a combination of pre and post editing in conjunction with midsurfacing is often the best approach.

In many cases, an extreme simplification of a system is all that is necessary to generate thermal results of the desired accuracy. For example, consider the blower shown in Figure 3-30. In this case, it is not desired to do a detailed analysis of the blower itself, but it is being included in a larger model where the heat generation and radiation of the bulk unit may be of importance.

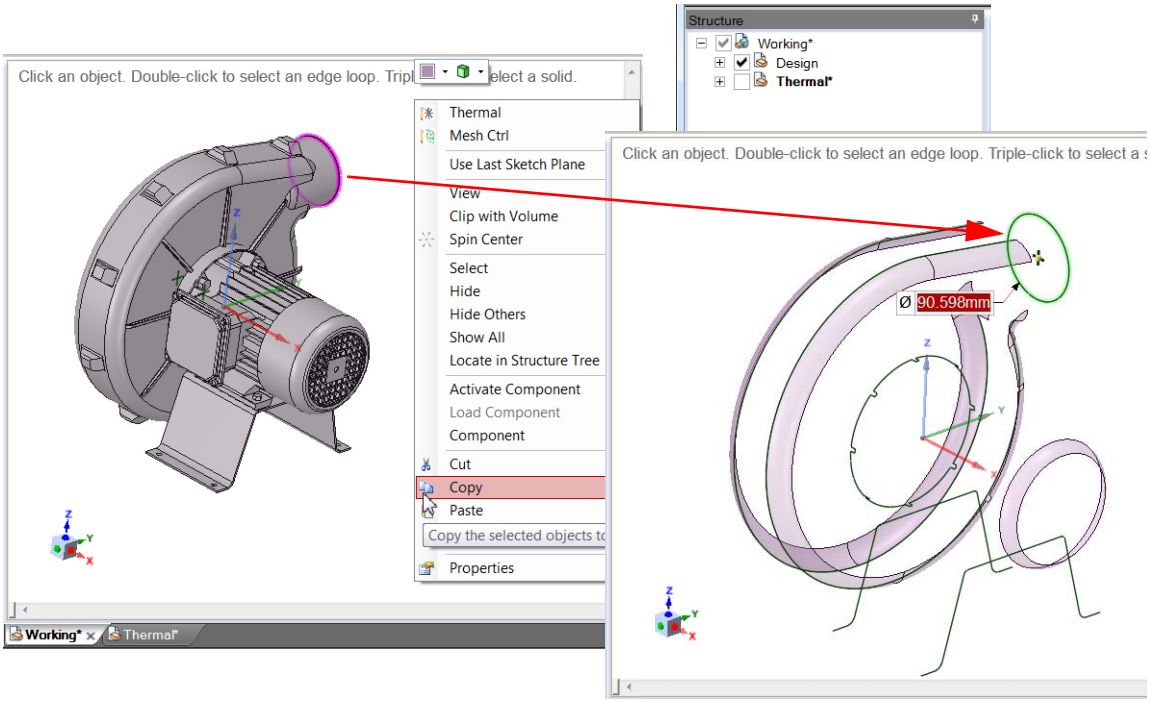


Figure 3-30 Key curves and surfaces can be copied from a detailed design and used to build a simpler representation

Individual entities of the detailed design, such as curves and faces, may be copied to a new component and used to generate appropriately simplified geometry. That component may be made external and then linked with Thermal Desktop.

A useful strategy is to keep the detailed design and the engineering geometry to be used for analysis in separate documents. Both of these documents can be part of separate master document. Desired components of the detailed design can be copied to the thermal document and modified there, as shown in Figure 3-30. This preserves the original imported model, and allows the engineering model to be visually compared to the original design model. Existing design and thermal documents can be inserted into a master document using the Insert > File command on the Design tab. Alternatively, in the master document create two new components, then make them external by right-clicking and choosing Source > Convert To External.

Link only the *thermal.scdoc* document to Thermal Desktop. Of course, use any meaningful names for the documents to organize the analysis effort. Once key curves and surfaces are copied from the design document into the thermal document, use any of the SpaceClaim editing and creation tools to complete the engineering model. For example, the +X and -X edges of the mounting bracket were copied to the thermal document. A blend using the Pull tool will create a surface between these two curves. The final simplification is shown in Figure 3-31

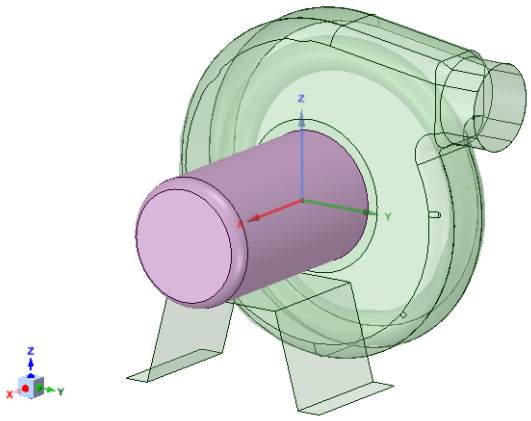


Figure 3-31 Simplified representation using a variety of SpaceClaim techniques

Many techniques and approaches are possible when simplifying complex geometry, there is no one single plug-and-chug formula. A knowledge of the features of SpaceClaim will greatly reduce the time spent preparing geometry for analysis. The user is encouraged to gain a basic understanding of the creation, repair, defeaturing, and simplification capabilities of SpaceClaim before attempting to generate a mesh for thermal analysis.

4 Meshing with TD Direct

TD Direct is an add-in to SpaceClaim that provides geometry import, mesh generation, and a dynamic, live link to Thermal Desktop. This section focuses on how to use the advanced mesh generation capabilities of TD Direct:

- Assembly meshing - multiple parts meshed together as a single mesh
- Auto-imprinting of surfaces at interfaces between parts to allow matched and merged meshes
- Meshing solid bodies and surfaces, as well as mixed dimensionality non-manifold objects
- Arbitrary unstructured triangular, quad-dominant, and quad surface meshes
- Arbitrary unstructured solid tetrahedral meshes
- Structured meshing of faces to produce quadrilateral elements
- Structured meshing of solids to produce hexahedral elements
- The use of face normals and edges to generate local material orienters for each solid and surface element
- Local mesh controls on parts, faces and edges
- Tagging of CAD bodies, faces, and edges to specify submodel names, property names, radiation analysis groups, and surface treatments (e.g., insulation)
- Tagging of CAD bodies, faces and edges to specify *domains* to be used for mesher actions, such as setting boundary nodes, or auto-generating Domain Tag Sets (sets of nodes, elements, and element edges that can be used to define conductors, contactors, heat loads, ties, and other network elements).
- Parametric definition of dimensions in the CAD model, and the ability to drive those dimensions with TD symbols. A symbol change will automatically generate a new mesh, with previously applied properties and connections retained.

4.1 Before Meshing

An attempt to directly mesh CAD geometry intended for the precise manufacture of an assembly will most likely be unsuccessful. Such geometry contains excessive detail, that even if successfully meshed, will produce an intractable model for analysis. SpaceClaim contains powerful features to prepare geometry for meshing. Unnecessary geometric details can be removed, and mid-surfacing operations can be performed to simply the model.

Often CAD geometry contains flaws that prevent meshing. To generate a mesh, the geometry must be topologically precise. That is, a volume should be bound by contiguous surfaces that share common curves and vertices. Some CAD data transfer formats are "light-weight" in that they permit topological inconsistencies as long as the geometry is visually acceptable. The geometry may contain missing or duplicate edges, gaps or missing faces. SpaceClaim also has the tools to repair defective geometry so that it is acceptable for meshing.

Effort to properly prepare the geometric model for meshing will be more than made up with easy mesh generation, and fast running thermal models. Engineering abstraction is still required to distill the detailed design model down into just the essential details to satisfy the requirements of the thermal analysis. SpaceClaim now makes this task almost trivial, without requiring the user to become a CAD expert.

More information, including sample models and tutorials, can be found at www.crtech.com/spaceclaim.html and www.spaceclaim.com.

4.2 Tags

Note: A limitation currently exists in that tag information will be lost if items are rearranged in the Structure tree. CRTech and SpaceClaim are working to resolve this issue and this limitation will be removed in the next release. As much as possible, please organize geometry in the Structure tree prior adding thermal and mesh control tags.

Tags are mark-ups of the geometry within TD Direct to specify information that will be needed by the meshing operation. "Information" can mean mesh controls, surface treatments, property names, submodel designations, and an association with a named *domain*, which itself has multiple purposes. These uses will be described later.

Tags can be defined within the current TD Direct document or inherited from a referenced document, called a *dependent document* in the SpaceClaim help documentation. Dependent documents contain *external* components. All tags that have been defined are displayed in the Tag Tree, which lists both tags defined in the currently active document as well as tags that have been defined in any dependent SpaceClaim documents. Dependent documents facilitate the editing and organization of an assembly model and is highly recommended as best practice, however, it is not necessary to create separate documents to construct an assembly.

Tags can be applied at any level of document dependency. Tags applied at higher levels overwrite those applied at lower levels. The tag tree represents the information that is sent to the meshing operation and to Thermal Desktop, and is the net result of any assembly tag overwriting. The tag tree displays information as Thermal Desktop will see it.

The tag tree is accessed by selecting the Tag tab in the SpaceClaim panels area (Figure 4-1). The tag tree panel is only available with TD Direct installed as a SpaceClaim add-in. The tag tree lists the tag types that have been assigned. Expanding a tag type shows the values that have been assigned. Clicking on a tag value will highlight and select those objects for which that tag value has been assigned.

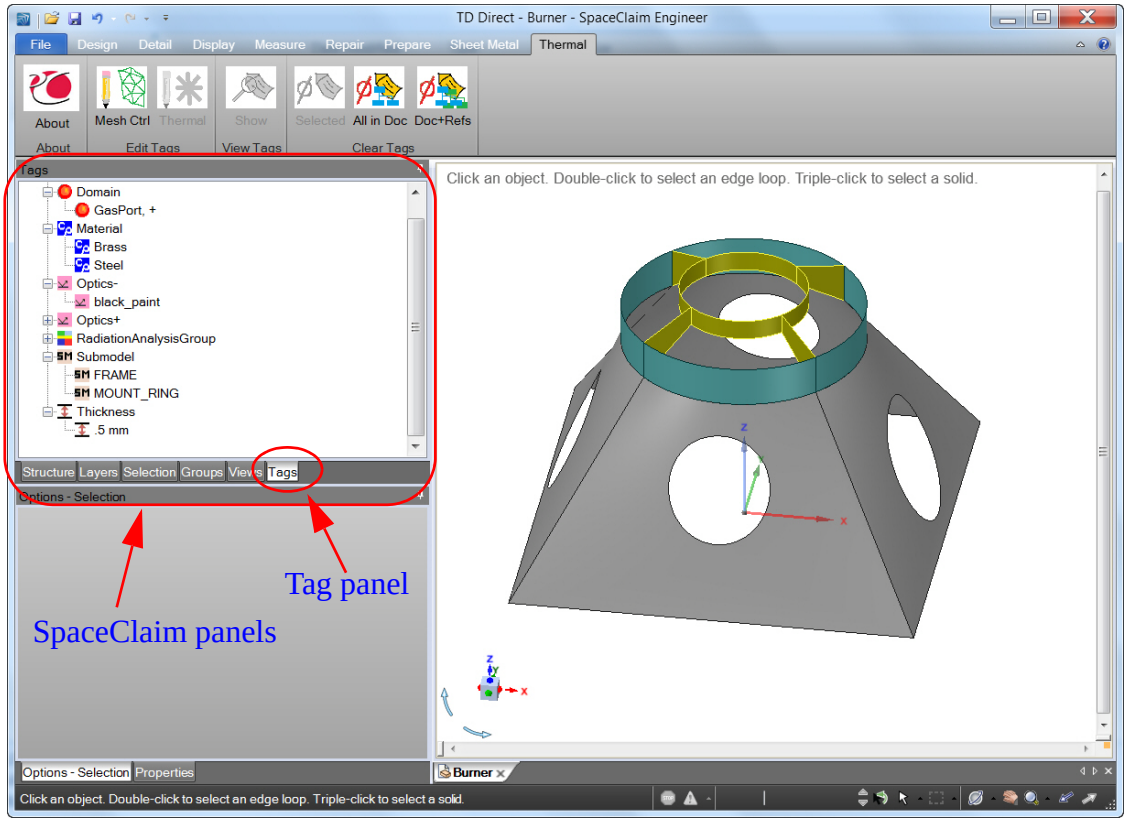


Figure 4-1 Tag tree panel shows information assigned to geometry

Tags are created by selecting one or more bodies, faces, or edges and choosing the icon on the Edit Tags group of the Thermal ribbon for the desired tag category. The two tag categories are Mesh Control and Thermal. A mesh control tag specifies local mesh controls for mesh generation, such as mesh density and element type (see Section 4.3). A thermal tag specifies information regarding the thermal model, such as optical and material property names, submodel names, or domains (see Section 4.4). The tag editing forms can also be accessed by right-clicking after a selection has been made.

To view a more detailed listing of the local and inherited tags assigned to an individual item, select that single item and then select the Show icon in the View Tags panel of the Thermal ribbon (Figure 4-2). The top pane of the form shows the tags that have been applied directly to the selected item in the current document. The lower pane shows the combined result of local and inherited tags defined in any referenced documents. The tag tree shows the net result of all tag applications in the entire assembly. The Show command allows an examination of the components of the assembly tree to see exactly at what level the tags were applied.

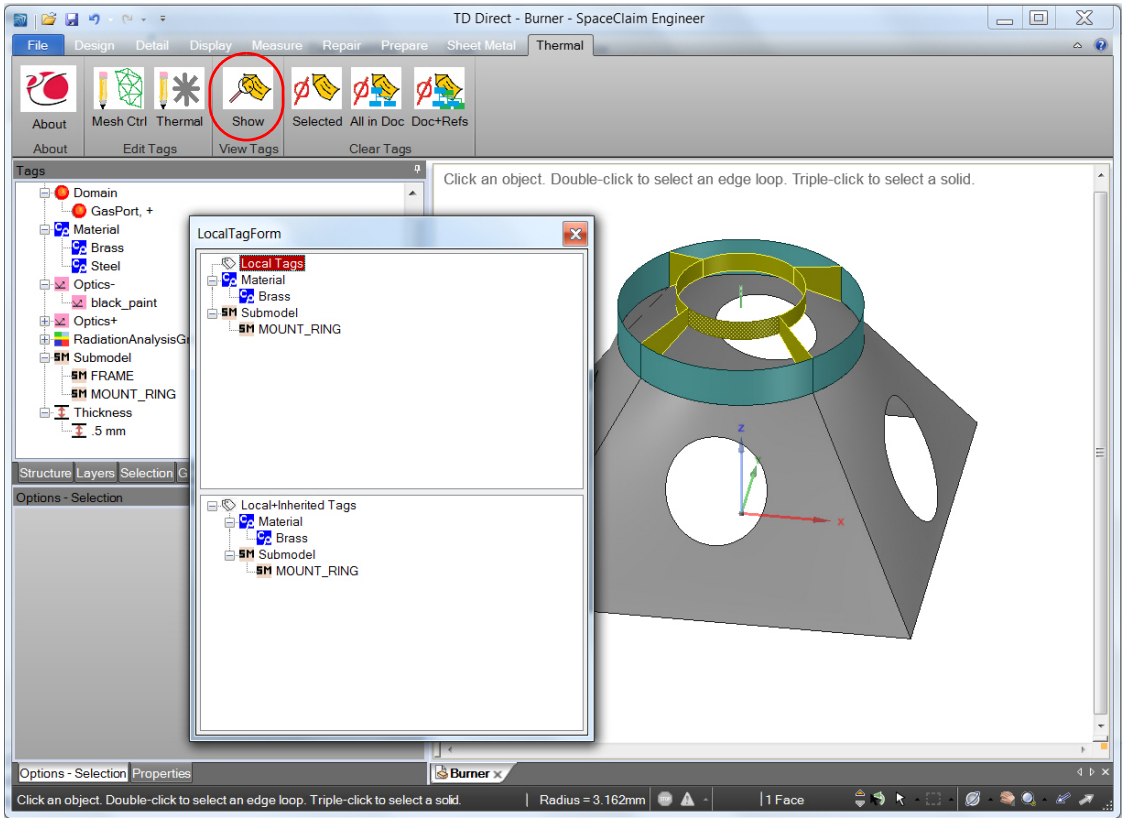


Figure 4-2 View local and inherited Tags

Deleting a tag will expose tags defined at dependent levels. If no dependent level tags are defined, deleting the tag will remove the attribute from the selected geometric entity. For example, to no longer generate a swept mesh on a volume region, remove the Number of Layers tag (see Section 4.3.6 for swept structured meshing).

Tags can be deleted in one of four ways:

- Deleting a value: edit the CAD entity, and delete the value in the appropriate field. Make a field blank to delete a value; do not use a zero. (Tags can't be deleted by selecting them directly in the tag tree itself. Instead, tags will disappear from the tag tree if they are no longer used by any CAD entity.) If a tag value is highlighted, all entities defining that value are selected. Invoking the appropriate tag editor will edit all entities simultaneously, and the tag can be changed or deleted.
- Deleting all tags for a selected item: select one or more items and select the Selected icon in the Clear Tags panel in the Thermal ribbon
- Deleting all tags in the current document: select the All in Doc icon in the Clear Tags panel in the Thermal ribbon
- Deleting all tags in the current document and all referenced documents: select the Docs+Refs icon in the Clear Tags panel in the Thermal ribbon

4.2.1 Length Units

All input fields that specify a length value for either the Mesh Control or the Thermal editors allow a value or an arithmetic expression, and an optional units specification. You can include the units specification as text in the input field, for example, “1.5 mm” instead of just “1.5” as the input. The units for each field may be specified independently.

If units are not specified, they are assumed to be the units assigned to the SpaceClaim document. SpaceClaim units may be set by clicking the SpaceClaim logo (or File tab) in the upper left hand corner and selecting SpaceClaim options.

4.3 Mesh Controls

Mesh controls in TD Direct define the mesh type and resolution. Mesh controls can also be used to exclude a part from meshing. The engineering geometry can be tagged using local mesh controls that override the default (global) mesh controls.

4.3.1 Default Mesh Controls

Default mesh controls are mesh controls that apply to any geometry without a mesh control tag. To set the default mesh controls, deselect all geometry by hitting <ESC> and select the Mesh Ctrl icon in the Edit Tags panel of the Thermal ribbon. The Default Mesh Controls form opens (Figure 4-3). In addition to the name of the window and the advanced options tab, the default mesh controls form differs from the local mesh controls form in color: the default mesh controls form is pale yellow and the local mesh controls form is light blue. The following subsections will define the mesh controls that are unique to the Default Mesh Controls form.

4.3.1.1 Ignore Small Features

On the Advanced Default tab of the Default Mesh Controls window, the first option is Ignore Small Features. Ideally, small features (those that will affect the mesh but have little effect on the thermal predictions) will have been removed from the geometry through simplification and repairs performed using the SpaceClaim healing and defeaturing capabilities. Occasionally, small features cannot be removed easily. In these situations, checking the Ignore Small Features option will mesh over features smaller than the specified value. Figure 4-4 shows the same geometry meshed with and without Ignore Small Features checked. The upper mesh, with Ignore Small Features unchecked, captures a small sliver in the geometry. The lower mesh, with Ignore Small Features checked, does not capture the small sliver in the mesh.

In addition, with Ignore Small Features unchecked, TD Direct will perform a check of edge length. A characteristic model size is computed from the largest edge length of a bounding box that contains the entire model. Any edge that has a length that is less than Small Edge Check Fraction times the characteristic length will be flagged and meshing

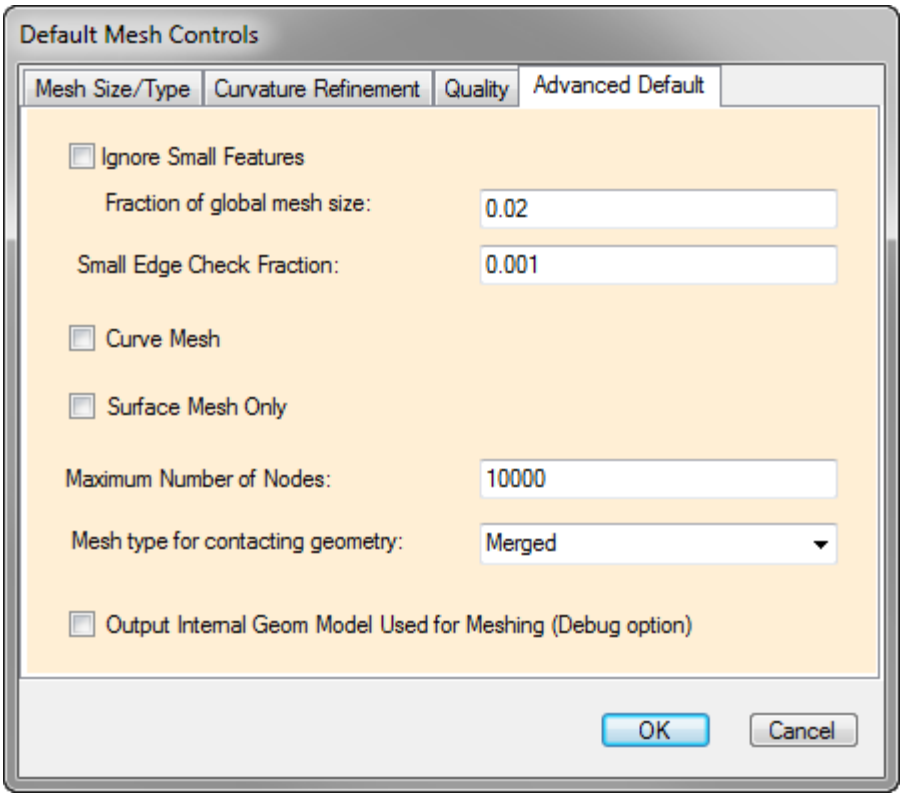


Figure 4-3 Default Mesh Controls form showing the Advanced Default tab

will not continue. Face areas are also checked and flagged. Any face area that is less than ten times the small edge check length squared will prevent meshing. Decrease this parameter to allow small edges and faces in the model to be meshed.

4.3.1.2 Curve Mesh

The use of Curve Mesh on the Advanced Default tab of the Default Mesh Controls window allows curved elements to be used instead of traditional straight elements. These curved elements can use fewer nodes and create a shape truer to the original geometry than traditional elements. Figure 4-5 shows two meshes created using the same parameters except the mesh on the left uses curved elements.

Radiation and contactor calculations may take longer for a curved element compared to a similar straight element, but representing the geometry with fewer elements may make the overall radiation and contactor computation time less. In addition, fewer nodes may make the final thermal network smaller and faster for SINDA/FLUINT execution. Where it is desired to have a coarse mesh on a non-planar surface, a few curved elements may be more efficient than many flat elements.

In most situations with curved geometry, using curved meshes is recommended.

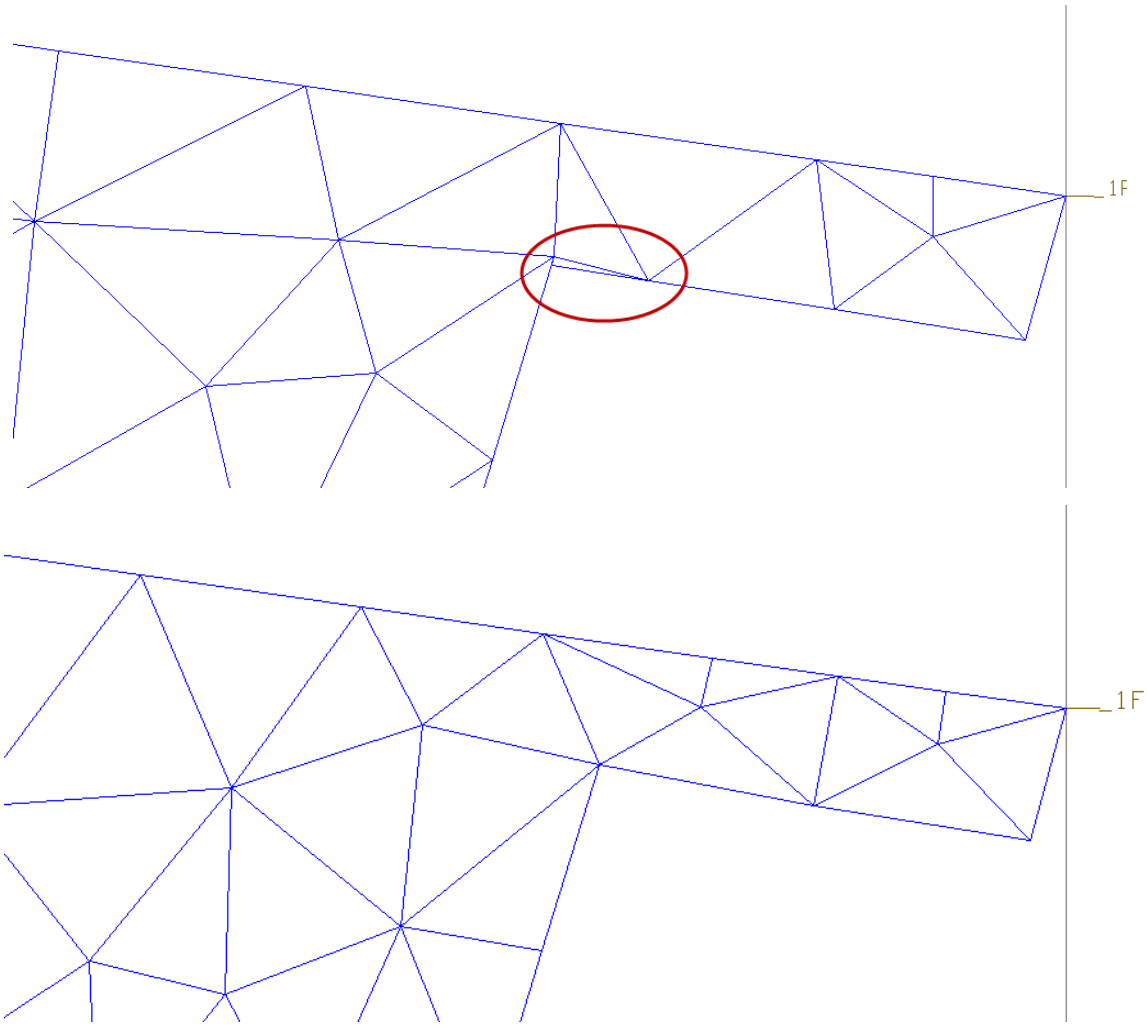


Figure 4-4 Mesh generated from geometry with a small feature (a sliver), with and without small features ignored.

4.3.1.3 Surface Mesh

TD Direct is optimized to work with solids, and some operations may be easier to perform on solids than surfaces. In many cases, however, the desired final product is a surface mesh. With this in mind, a user can create the geometry as a solid, and then select Surface Mesh on the Advanced Default tab of the Default Mesh Controls window. This will create a mesh of the faces of the solids and not the solids, resulting in only 2D elements. For example, a solid pipe may be created and manipulated in TD Direct, but the desired mesh may consist of only the outer surface, and the thickness of the pipe can be defined abstractly on the surface. In this case, the inner pipe wall and the rings on the end would be surfaces that are not wanted in the mesh. See Section 4.3.2.2 to see how to exclude individual surfaces from this operation.

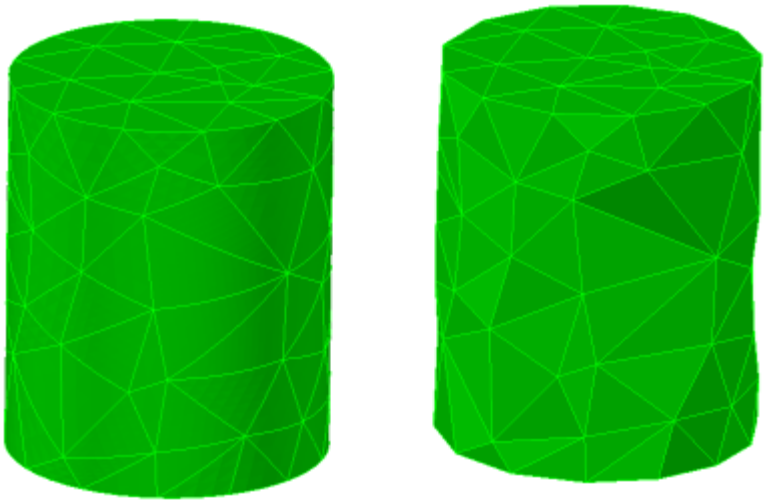


Figure 4-5 Curved and Traditional Meshes

4.3.1.4 Maximum Number of Nodes

The Maximum Number of Nodes field, on the Advanced Default tab of the Default Mesh Controls window, allows the user to limit the number of nodes in the generated mesh. If the mesh generates more nodes than specified in this field, the mesher will halt and will not generate the mesh in Thermal Desktop. This is a safeguard against inadvertently setting a mesh control that otherwise might generate an intractably large mesh and consume excessive CPU and memory resources.

4.3.1.5 Mesh Type for Contacting Geometry

The Mesh type for contacting geometry pull-down menu determines how the mesh behaves when two parts of an assembly (components in the SpaceClaim documentation) are in contact (sharing faces or edges).

The options are:

- Merged: the two faces are imprinted with each other and the meshes are aligned at the interface; the nodes are merged to form a contiguous thermal body
- Independent: the meshes are completely independent of each other
- Matched: the two faces are imprinted with each other and the meshes are aligned; the nodes are *not* merged, leaving the thermal connection to be left open or completed with a contactor between the two surfaces. The matched mesh option allows the user to economize on the contactor calculations by setting the contactor integration intervals to 1 (see Thermal Desktop User's Manual).

A merged mesh applies to all interfaces in a document. If a document contains some interfaces that must be merged and some interfaces that must be matched, choose the matched option globally (as the default). Individual interfaces can then be merged using the Merge if imprinted local mesh control tag.

4.3.1.6 Output Internal Geometry Model Used for Meshing

This option is for trouble-shooting a mesh. Use this option upon request by technical support. When enabled, a special file will be produced that contains geometry and mesh control assignments. In the event of a meshing failure, it will also make visible the intermediate SpaceClaim document that is the result of imprinting and merge operations. This internal document is normally deleted upon successful meshing. When displayed, any geometric items related to a meshing failure will be highlighted. It is not necessary to save the intermediate model unless requested by technical support.

4.3.2 Local Mesh Controls

Local mesh controls are tags applied to specific geometrical items such as bodies, faces, or edges. A local mesh control tag is created by selecting the desired item or items and then clicking the Mesh Ctrl icon in the ribbon. The Local Mesh Controls window opens (Figure 4-6). The local mesh control window differs from the default mesh control window in color: the local mesh controls window is blue where as the default mesh controls window is yellow. Also, the names of the windows are different, and the Advanced Local tab appears only in the local mesh control form.

The following subsections will define the mesh controls that are unique to the Local Mesh Controls form. Local mesh controls can only override the default controls as needed to refine the mesh. For example, larger elements cannot be specified along an edge with a local control if the default control specifies a smaller element. In other words, local mesh controls can only result in a finer mesh, not a coarser one.[1]

4.3.2.1 Merge and Merge Priority

The Merge if Imprinted option is a local control for merging the nodes on touching surfaces if the advanced default option mesh type for contacted geometry (Section 4.3.1.5) is set to *Matched*. When nodes are merged, the thermal tags (Section 4.4) of the item with higher Merge Priority are used for the interface, and those with lower priority are not used. To ensure that the tags of a particular item *are* used, set the merge priority for that item to be an arbitrarily large number, which is at least larger than the merge priority of other surfaces at the same interface.

1 This does not mean there is no value to specifying a larger *relative* size at the local level than at the global (default) level. The reason is that the characteristic (bounding box) dimension of the local object will be smaller than the characteristic dimension of the whole drawing. For example, if the global relative size is left at the default of 0.1, a local relative mesh size of 0.25 may result in a smaller element size on that local object if it is smaller than 0.1/0.25 = 40% of the rest of the drawing.

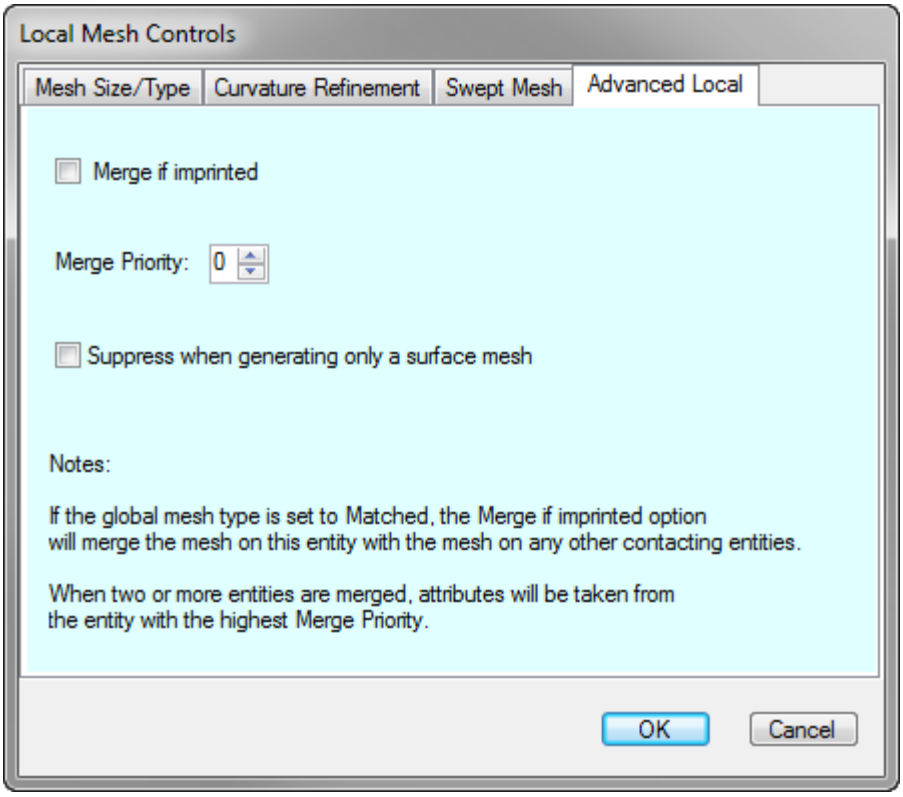

Local Mesh Controls
Mesh Size/Type
Curvature Refinement
Swept Mesh
Advanced Local
Merge if imprinted
Merge Priority:
0
Suppress when generating only a surface mesh
Notes:
If the global mesh type is set to Matched, the Merge if imprinted option
will merge the mesh on this entity with the mesh on any other contacting entities.
When two or more entities are merged, attributes will be taken from
the entity with the highest Merge Priority.
OK
Cancel

Figure 4-6 Local Mesh Controls window with the Advanced Local tab open

4.3.2.2 Suppress When Generating Only a Surface Mesh

Suppress when generating only a surface mesh is a tag that can only be applied to surfaces. If the Surface Mesh (see Section 4.3.1.3) option is used, it is likely that not all surfaces should be meshed. For example, a user may have created a solid cylinder to represent a hollow pipe, and thus the circular end caps of the cylinder should be suppressed. If the Surface Mesh option is not used, this tag has no effect.

4.3.3 Mesh Size and Type

As documented in the previous two subsections, the mesh size and type controls can be set as default global values or as local tags. There are a few differences between available options at the default level and at the local level. Unless otherwise indicated, the settings in the following subsections can be applied as either default values or as local tags.

To demonstrate mesh size and type settings, the switch box geometry shown in Figure 4-7 will be used. The switch box is an open-bottomed box with a flange around the base. For this discussion, the box will be modeled with solid elements so as to capture temperature gradients through the thickness.

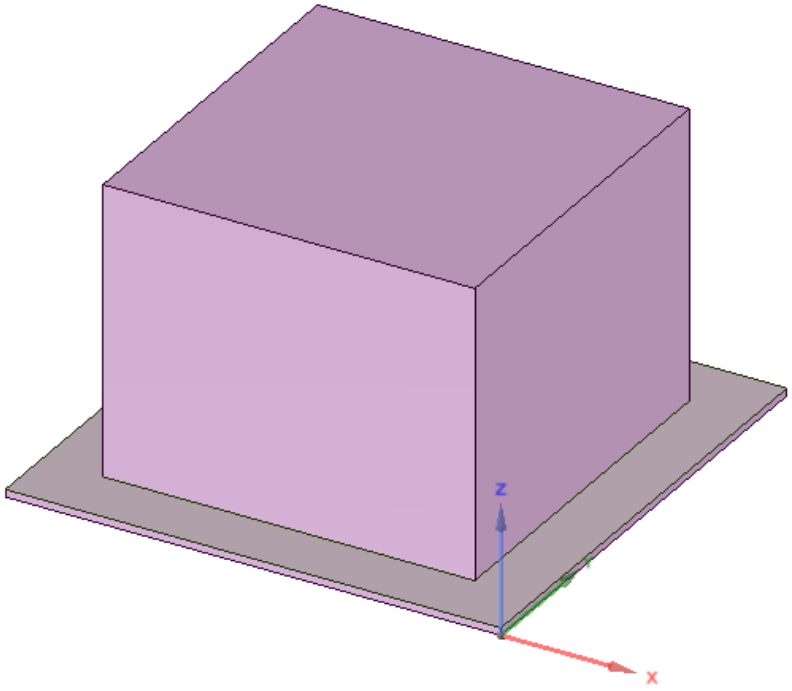


Figure 4-7 Switch box geometry

4.3.3.1 Mesh Size

Mesh size determines the size of the element edges. Solid element sizes can be set directly, but are often specified indirectly as a function of surface and edge element sizes, which will override solid settings if they exist. Therefore, this control parameter largely applies to surfaces and edges.

The size can be specified as an absolute value (a physical length) or a relative value (a fraction of the maximum dimension of the selected geometry). The mesh size is set on the Mesh Size/Type tab of the Default Mesh Controls or Local Mesh Controls windows (Figure 4-8).

The recommended method for setting mesh sizes is to set the maximum allowable mesh size in the default settings and then refine the size near appropriate portions of the geometry using local controls.

The relative mesh size can be any value between 0 and 1, inclusive of 1. Values less than 0.01 are discouraged since they will drive up computation costs. Small values may cause the number of nodes to exceed the maximum number of nodes value (Section 4.3.1.4) set in the default mesh controls.

The following figures show example meshes with different mesh size settings:

- Figure 4-9 shows the switch box mesh with a default relative mesh size of 1.
- Figure 4-10 shows the switch box mesh with a default relative mesh size of 0.1

- Figure 4-11 show the switch box mesh with a default relative mesh size of 0.5 and a local mesh control size of 0.1 along the flange edge faces (shown by the black lines)

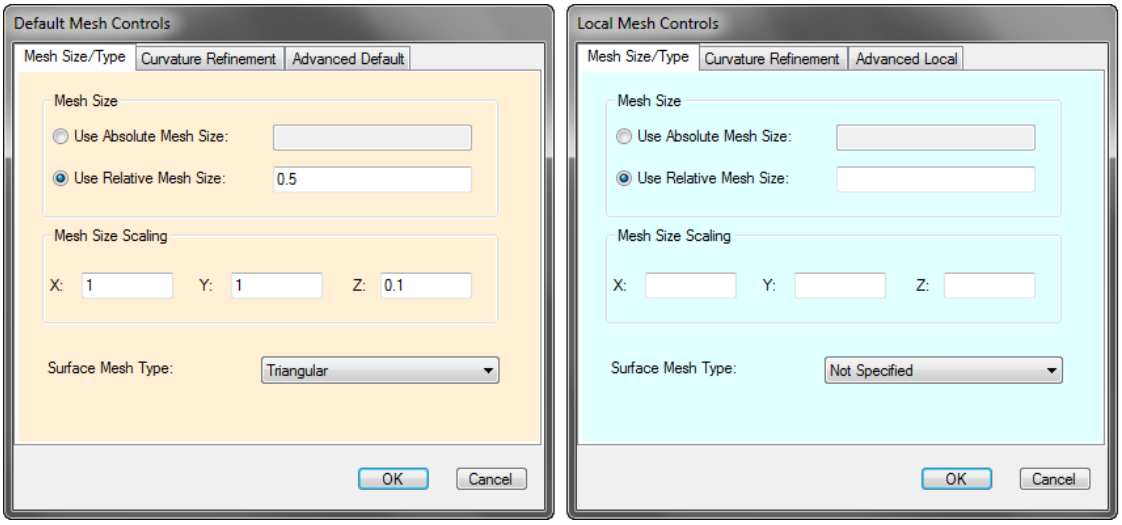


Figure 4-8 Mesh Size and Type tab on Default and Local Mesh Controls windows

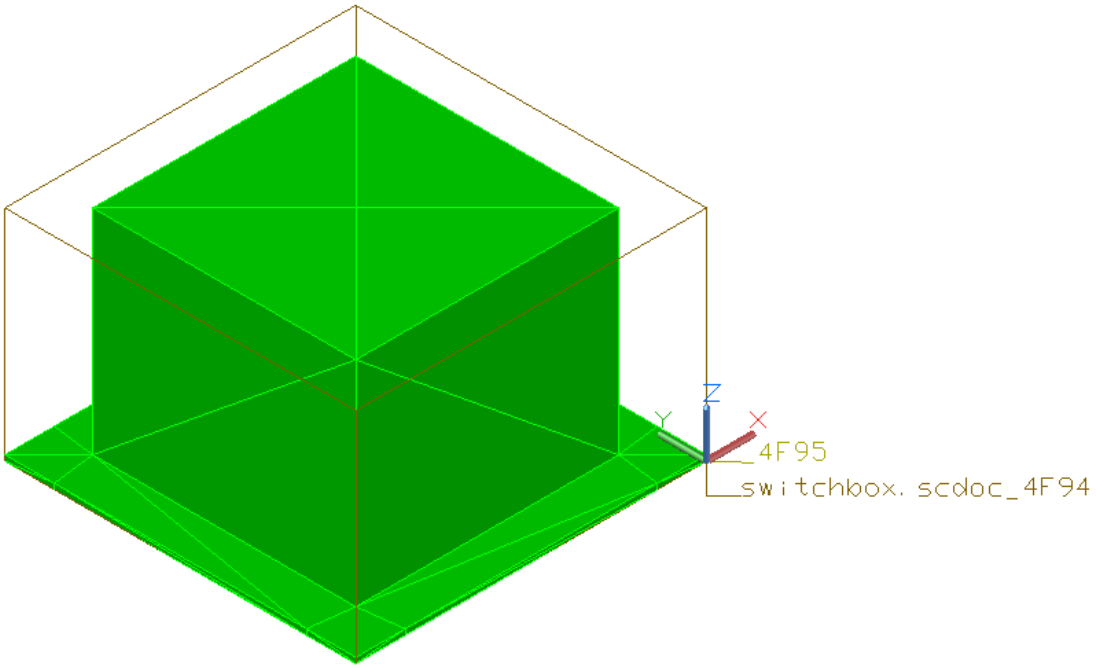


Figure 4-9 Switch box with default relative mesh size set to 1

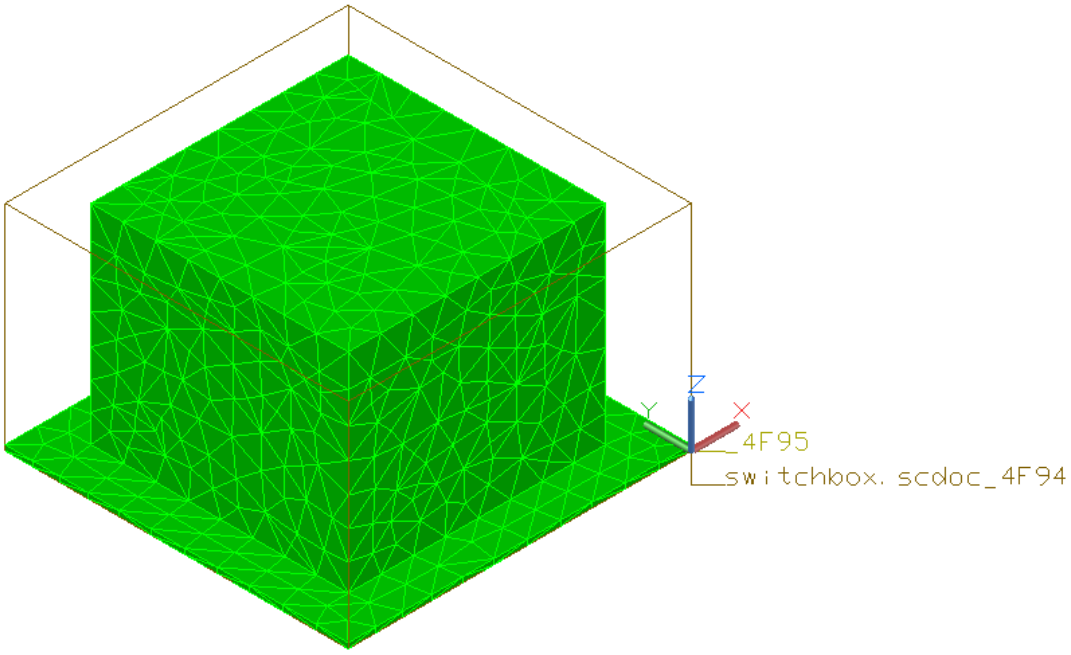


Figure 4-10 Switch box with default relative mesh size set to 0.1

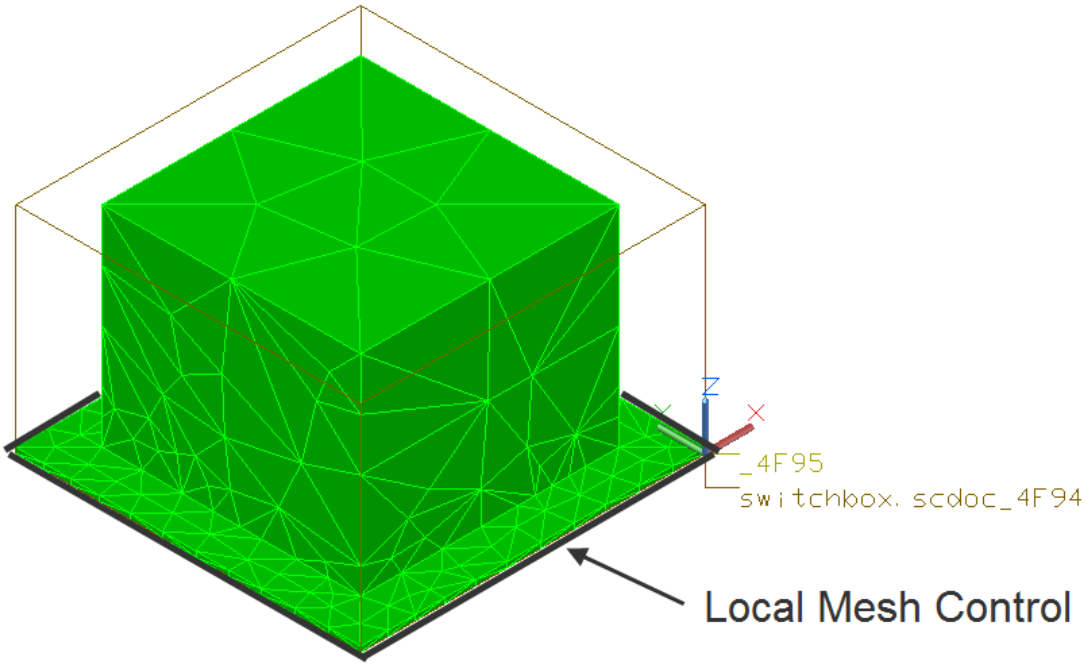


Figure 4-11 Switch box with default relative mesh size set to 0.5 and a local mesh size along the flange edges set to 0.1

4.3.3.2 Mesh Size Scaling

Mesh size scaling allows specifying a different mesh size in each direction of the SpaceClaim coordinate system. Valid values are between 0 and 1, inclusive of 1.

Figure 4-12 shows an example of mesh size scaling in the Z direction. Note the tighter mesh pattern in the vertical direction.

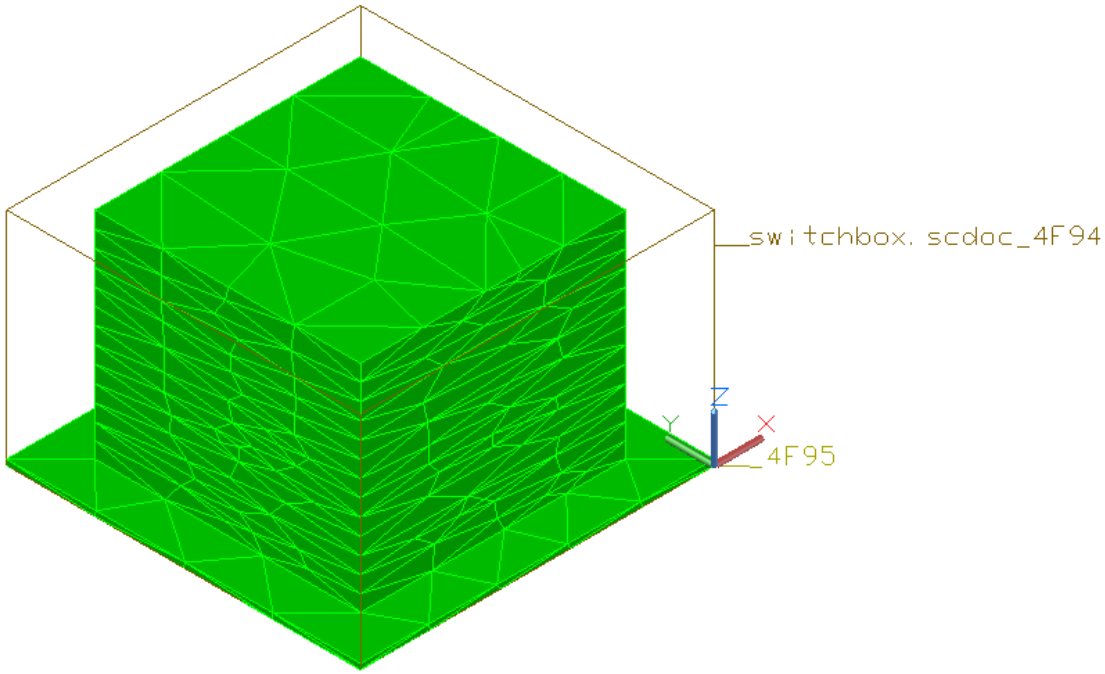


Figure 4-12 Switch box mesh with default relative mesh size set to 0.5 and the default mesh size scaling set to 0.1 in the Z direction.

4.3.3.3 Surface Mesh Type

The Surface Mesh Type option specifies the type of elements that will be used on the surface of the mesh. The available surface mesh types are:

- Triangular: the surface mesh is made up entirely of triangular elements (Figure 4-12)
- Quad Dominant: the surface mesh will be primarily quadrilateral elements, but triangular elements may be used (Figure 4-13)
- Quadrilateral (not available as a default setting, just as a local override): all surface elements are quadrilateral elements (Figure 4-14).

The solid elements types are automatically determined by the mesher to satisfy the mesh settings and the surface element types. In the figures referenced above, the surface mesh types were set in the local mesh controls with the entire solid selected.

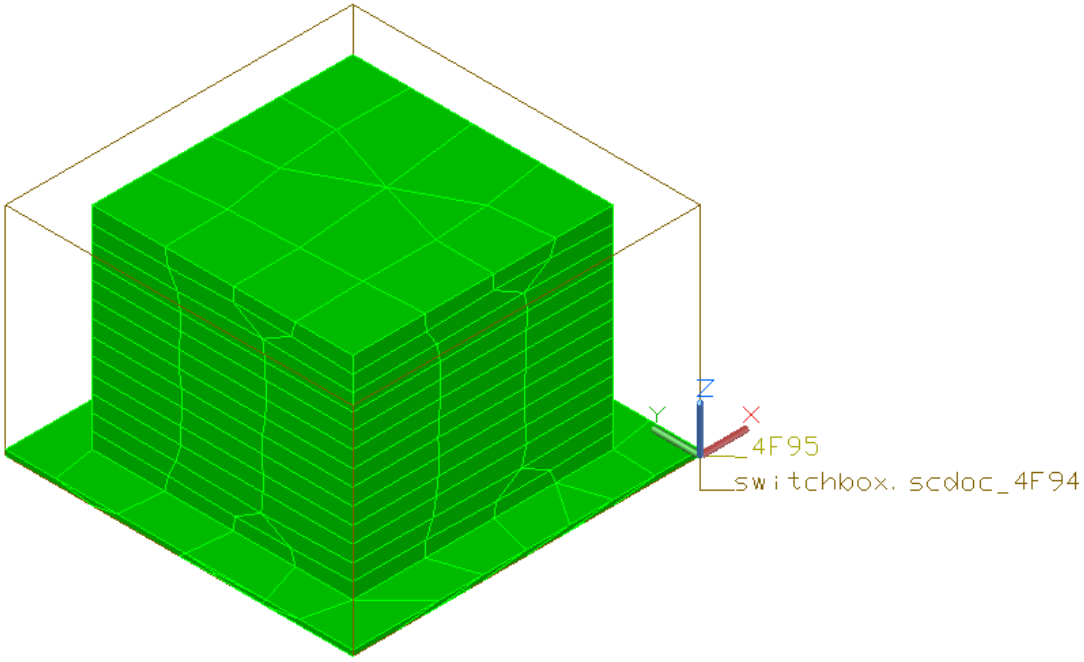


Figure 4-13 Quad Dominant mesh with default relative mesh size set to 0.5 and the default mesh size scaling set to 0.1 in the Z direction.

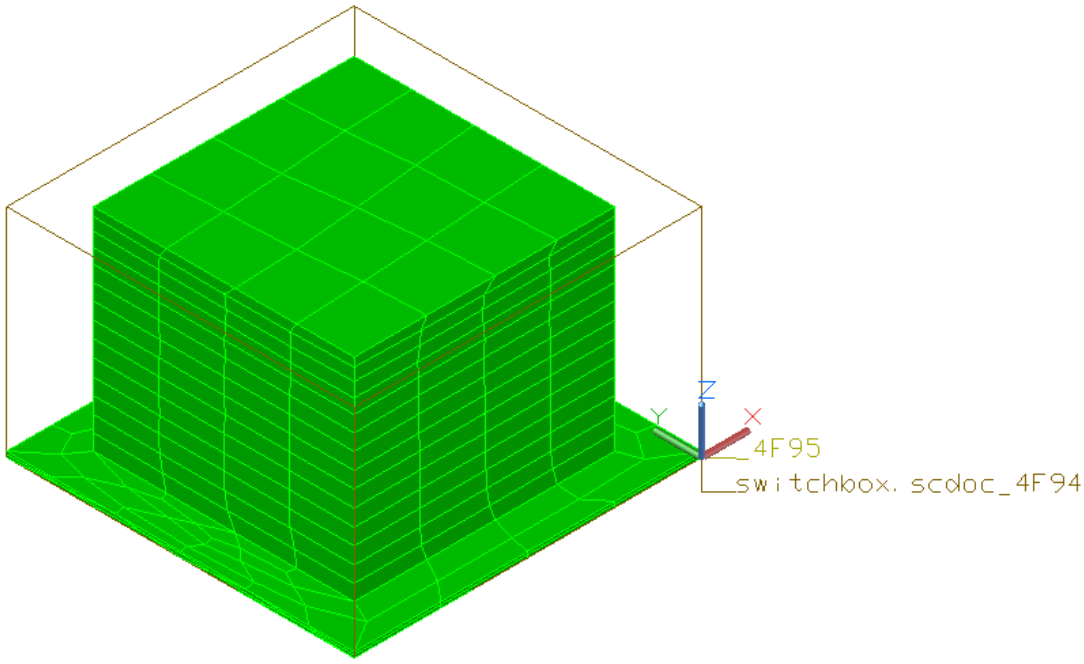


Figure 4-14 Quadrilateral mesh with default relative mesh size set to 0.5 and the default mesh size scaling set to 0.1 in the Z direction.

4.3.4 Curvature Refinement

Curvature refinement allows meshes to be more refined along curves in order to better capture the shape of the curve. The curvature refinement settings are found on the Curvature Refinement tab of the mesh controls windows (Figure 4-15).

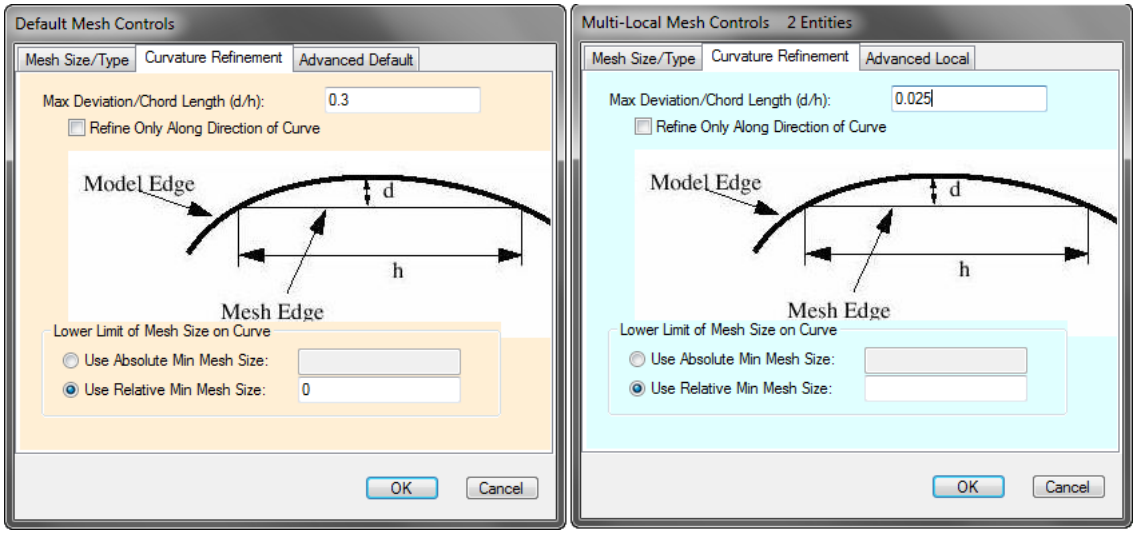


Figure 4-15 Curvature Refinement tab on Default and Local Mesh Controls windows

The specified value for curvature refinement is the maximum deviation divided by the chord length. This relationship is graphically described on the mesh controls form as seen in Figure 4-15.

The Refine Only Along Direction of Curve checkbox limits the curve refinement to the curve direction when checked. This option allows the mesh to be coarser in the direction perpendicular to the plane of the curve.

The values for the Lower Limit of Mesh Size on Curve option prevent the mesh from becoming too small on curves with extremely small radii.

4.3.5 Mesh Quality

The mesh quality controls are global controls that can be adjusted by the user. These controls are shown in Figure 4-16.

Mesh Size Transition Speed indicates how quickly the size of the elements can grow as the elements get farther from an area where more dense nodes were required. Figure 4-17 shows two meshes of the same object. A dense local mesh control was applied on one end of the objects. The only difference between the meshes is that the mesh on the left used the fastest Mesh Size Transition Speed and the mesh on the right used the slowest. The slower transition speed results in a higher quality mesh, but at the cost of more nodes. This transition speed control will be used in the creation of the initial mesh creation regardless of the quality slider setting.

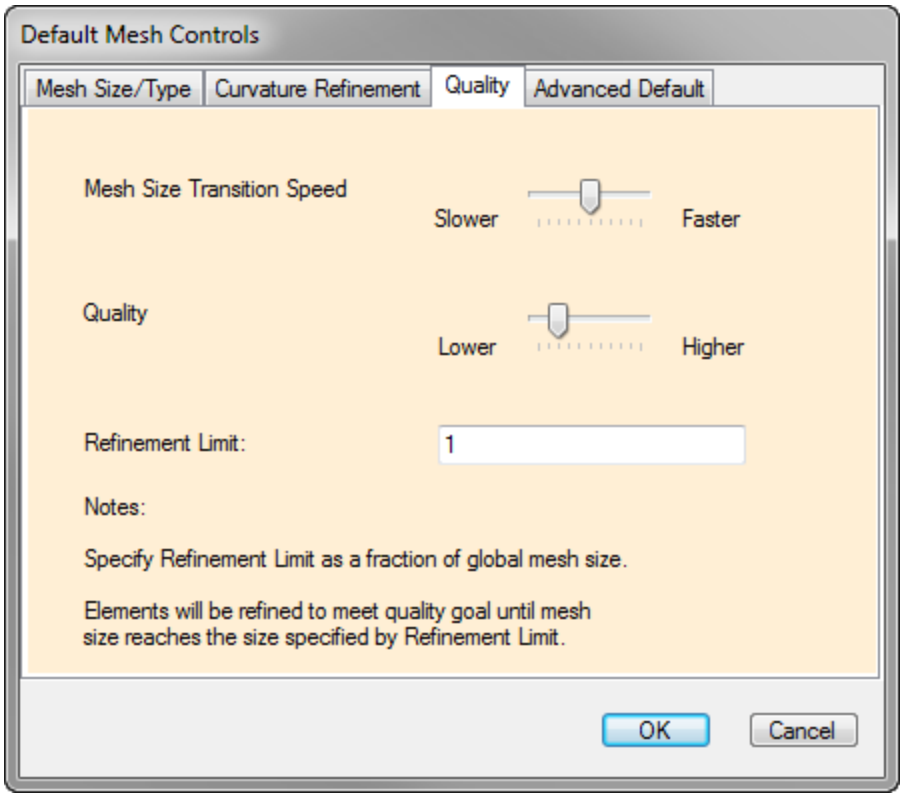


Figure 4-16 Mesh Quality Controls

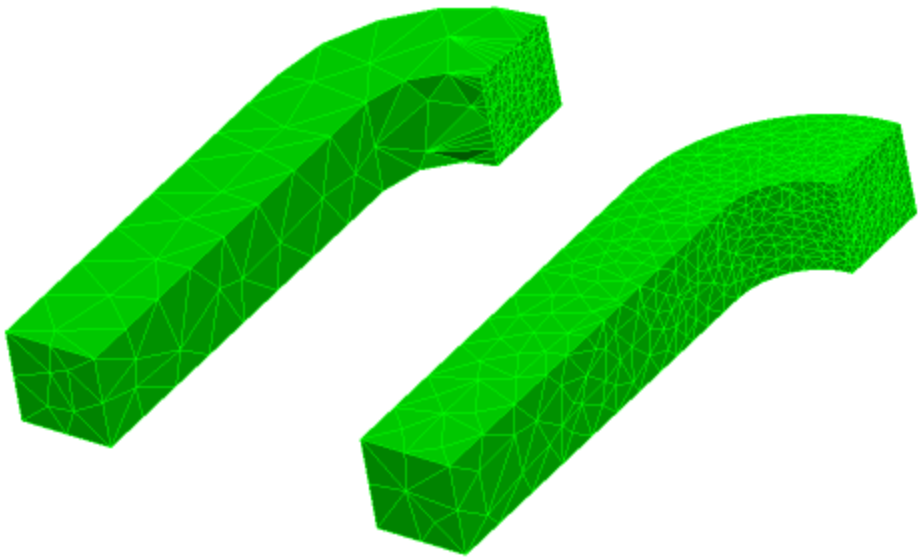


Figure 4-17 Fast and Slow Mesh Size Transition Speeds

Quality can be raised or lowered using the slider. The quality of the mesh is measured after an initial pass, and it will then be refined based on the position of the slider and the Refinement Limit. The Refinement Limit is a fraction of the global mesh size that estab-

lishes a lower size limit on elements as they are adjusted to improve mesh quality. Therefore, a Refinement Limit of 1 means that no mesh adjustments will be made for quality as the element size cannot be reduced. A Refinement Limit of 0.5 means that elements can be reduced down to half of the global mesh size to meet the quality requirements.

For more information about the quality of the mesh in Thermal Desktop, see the Thermal Desktop User's Manual (RCCHECKELEMENTS).

4.3.6 Structured Meshing

A structured mesh contains elements that are aligned with the principle parametric directions of the surface or solid that is being meshed. In contrast, an unstructured mesh arbitrarily paves a surface with triangle or quadrilateral elements, and fills an arbitrary volume with tetrahedral or pyramid elements. Structured meshes can be created by sweeping an edge across a face, or by sweeping a face across a volume. Sweeping an edge across a face produces quadrilateral elements. Sweeping triangles across a volume creates wedge elements, and sweeping quadrilateral elements across a volume creates hexahedral elements.

A face that is tagged to create a swept mesh can also be used to create a swept volume mesh. In this case, an all-hexahedral mesh is created. A quadrilateral mesh is first created on the face by sweeping between two edges, and then this mesh is swept between two faces on the solid to create a hexahedral mesh. Figure 4-18 shows an example of some swept structured meshes.

Swept volume meshes may contact each other, and if possible, the mesh will be adjusted and imprinted on adjoining faces to complete the meshing operation. Figure 4-18 shows an example of two overlapping slabs, each of which is a swept volume. The mesh is automatically matched where the slabs overlap.

Local mesh controls are applied to design geometry to create a structured mesh. To create a swept mesh on a face, an edge must be selected and tagged as the "source" edge. An edge on the opposite side of the face must be selected as the "destination" edge. The face is tagged with an integer number of "layers". A non-blank value for a face indicates that it is desired to create a structured mesh on the face. The face must contain one and only one edge marked as the source edge, and one and only one edge marked as the destination edge.

The swept face mesh is created by sweeping from the source edge to the destination edge across the face. Global and/or local mesh size and curve refinement parameters are used to first create the vertices along the source edge. These line segments are then swept and copied to create the "number of layers" of quadrilateral elements.

Likewise, to generate a swept volume mesh, the Number of Layers tag is applied to a solid design body. One or more faces on the body must be specified as source faces, and one or more faces must be specified as destination faces. Swept volume meshes are not restricted to a single source and destination face, however, other restrictions apply that will be explained later.

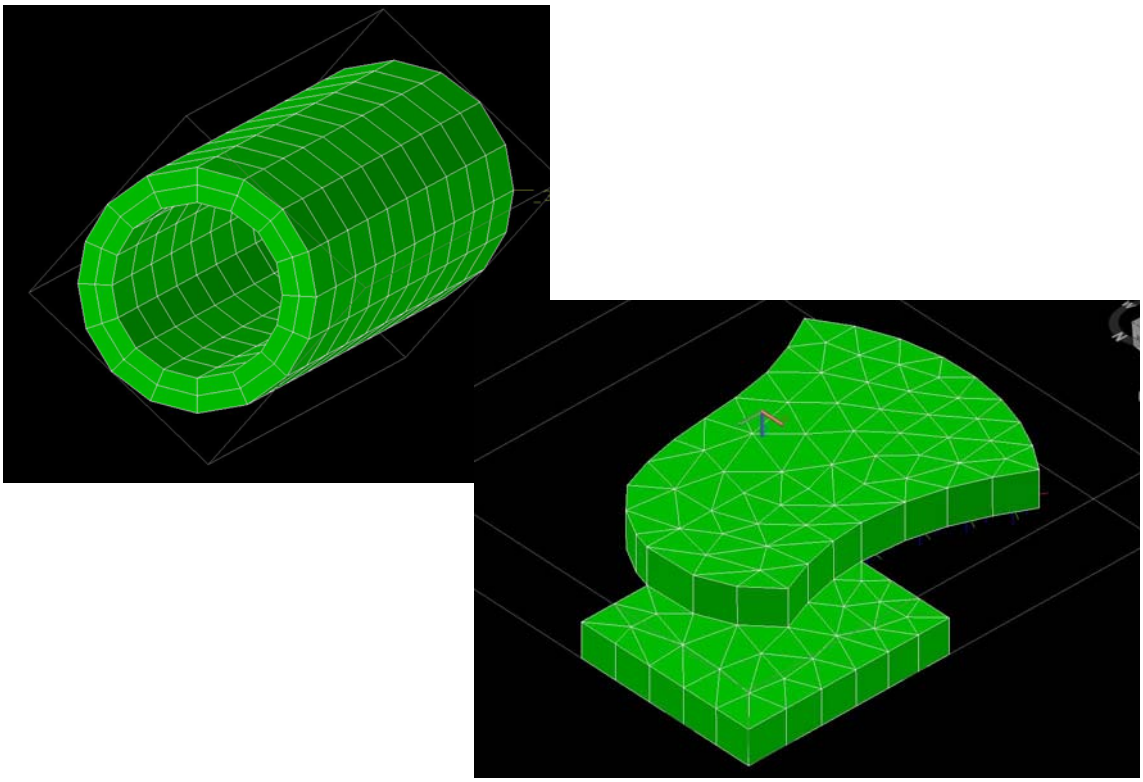


Figure 4-18 Examples of structured meshes created by sweeping edges and faces

The local mesh control dialog that is used to specify parameters for swept meshes is shown in Figure 4-19. The top portion of the form is used on the entities for which a swept mesh is desired. Entering a non-blank value for Number of Layers will cause a structured mesh to be created.

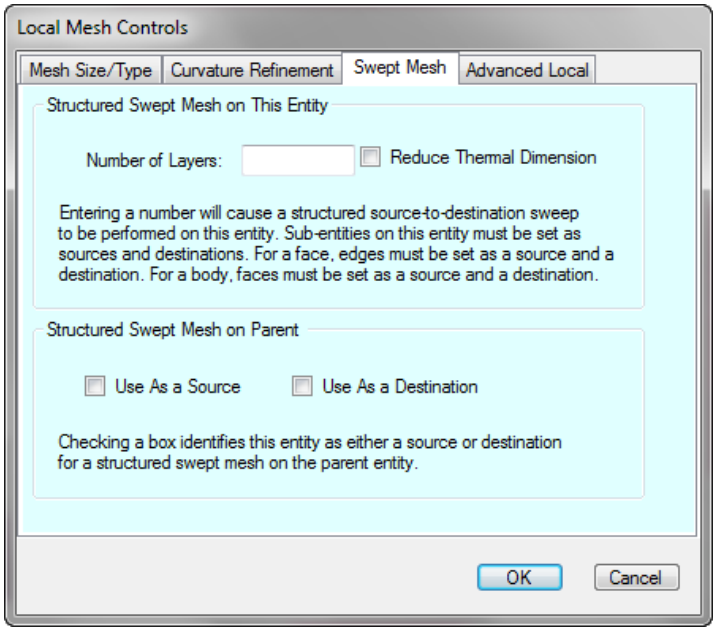

Local Mesh Controls
Mesh Size/Type
Curvature Refinement
Swept Mesh
Advanced Local
Structured Swept Mesh on This Entity
Number of Layers:
Reduce Thermal Dimension
Entering a number will cause a structured source-to-destination sweep to be performed on this entity. Sub-entities on this entity must be set as sources and destinations. For a face, edges must be set as a source and a destination. For a body, faces must be set as a source and a destination.
Structured Swept Mesh on Parent
Use As a Source
Use As a Destination
Checking a box identifies this entity as either a source or destination for a structured swept mesh on the parent entity.
OK
Cancel

Figure 4-19 Local mesh controls to create a swept mesh on a face or volume

The lower portion of the form is used to specify the source and destination entities. These are sub-entities of the entity for which the swept mesh is being generated. Source and destination edges must be specified for a surface, and source and destination faces must be specified for a swept volume.

The Reduce Thermal Dimension option is available for swept volumes or surfaces. If this option is checked, the number of layers will default to one, and the same node ID's will be used on both the source and destination faces or edges. For a solid, this essentially makes the three dimensional finite element network appear to be two dimensional to SINDA/FLUINT. Thin solid structures may be meshed using a single layer of three dimensional elements, and effectively appear to be two dimensional to the solution process. Only lateral, in-plane conduction terms will be produced. Using this option from edge to edge across a surface similarly removes the gradient across the surface.

When using Reduce Thermal Dimension, thickness information is retained, and the thickness does not need to be constant over the swept object. Reduced dimension meshes will retain a one-dimensional through thickness network, even when stacked one upon another. This method may be easier than performing mid-surfacing operations on solids, and retains the 3D geometric structure to facilitate contact with other portions of the thermal model.

4.3.6.1 Swept Mesh Limitations

Swept faces must have a single source edge, and a single destination edge. Swept volumes will use one of two algorithms to create a mesh depending on the number of source and destination faces. If a swept volume has a single source face, and a single destination face, it will use the "general" sweep algorithm. If the swept volume contains more than one source or destination face, either by the user or an imprint operation, then the "thin section" algorithm will be used. If the geometry is not amenable to thin section meshing, it can always be split onto smaller, more regularly shaped bodies and meshed with the general algorithm.

Some limitations of the general algorithm:

1. The source and destination faces should not differ greatly geometrically.

2. Every lateral face (model faces along the sides of the sweep) may only have a single loop (i.e no interior loops, holes, etc.).

3. Each lateral face must have all of its edges roughly parallel to or orthogonal to the sweep direction.

4. The destination face of an sweep may not be a lateral face of another sweep (but the source face may be).

5. The source and destination face must share a region (the region that will be swept).

6. Attributes cannot be set such that the sweeps form a closed loop.

The general algorithm can be used for stacked bodies; however, the geometry of the destination face of one body must match the source face of the adjoining region. Consider the model shown in Figure 4-20.

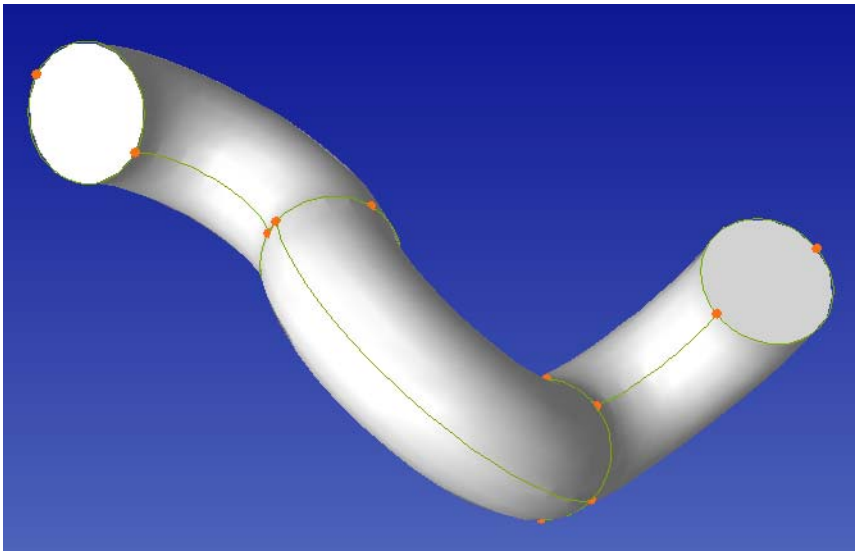


Figure 4-20 Multi-section model with non-matching vertices

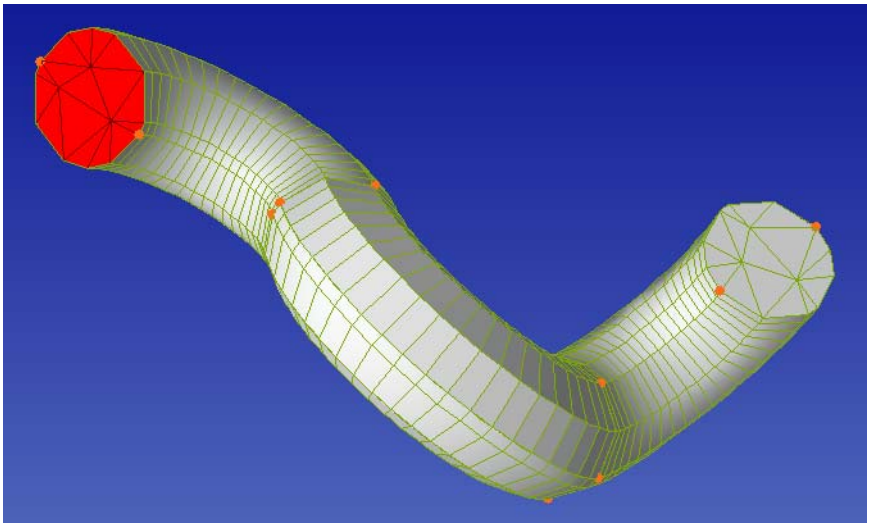


Figure 4-21 Swept volume mesh of model

The model consists of three elbow regions. Due to the positioning of the bodies, the source and destination faces for sweeping the regions do not match. Take for instance the left most face of the model. It contains two model vertices. This face will sweep through the first region to a destination face that contains four model vertices.

The general sweep volume mesher works by placing corresponding mesh vertices on the source and destination faces. In this first region, the two extra model vertices from the destination are mapped to mesh vertices on the source face's edges. In the case of serial sweeps such as with this model, all model vertices are mapped to all source/destination edges.

Thin-section meshing is a specialized type of swept volume meshing. It is used to create a structured mesh with multiple layers through the thickness of the thin-section. The feature differs from general swept volume meshing, in that, the set of source and destination faces do not require a one-to-one mapping. Multiple faces can be specified for the source and destination, and thin sections can be stacked that do not exactly line up, as shown in Figure 4-22

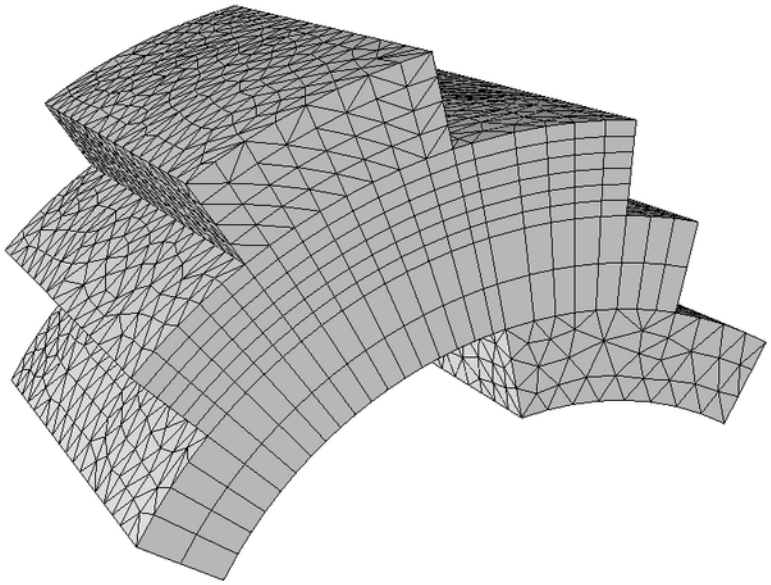


Figure 4-22 An example of stacked thin section swept volume meshes

Thin section regions can be coupled (or stacked) together, but must obey the following:

1. A face cannot be a source of more than one thin-section body

2. A face cannot be a destination of more than one thin-section body

3. A face cannot be both a lateral and a cap (source or destination) of two thin-section bodies.

4. A face may be a source and destination of two thin section bodies.

5. The thin-section coupling cannot be cyclic. There must be a single source and destination set. Closed items can be split to allow thin-section meshing.

6. The entire model region must be thin in one direction.

7. The body must topologically be a thin block. e.g L-shaped or irregular shaped regions are not allowed.

8. The lateral (side) faces, if any, must be rectangular or annular. No internal holes or non-manifold edges are allowed on lateral faces.

9. Geometrically, the body must be locally orthogonal. That is, the distance of a bi-directional projection of a point from source to destination, back to source must be much less than the mesh size specified on the source face.

10. The source and destination faces must be unstructured. That is, a swept face mesh is not allowed on a source or destination face for a thin section. Split the solid such that there is only one source and one destination face if a swept face is desired to be swept across a volume.

In summary, the general algorithm is chosen if single source and destination faces are specified. Thin-section meshing is performed if more than one source or destination face is chosen, or additional faces are created due to a merge or imprint operation from adjoining bodies. If the thin-section meshing is unsuccessful, perhaps because the body is not locally orthogonal, it can often be split such that the subsections can be meshed with the general algorithm.

4.4 Thermal Tags

Thermal tags are used to mark the engineering geometry with information that will be used in constructing the thermal model. Information such as material properties and submodels are assigned to finite elements solids. Thicknesses, optical properties and radiation groups are assigned to finite element surfaces. A special type of thermal tag, called the Domain tag, can be used to specify groups of entities for later use in Thermal Desktop.

The term “thermal” is used to distinguish this type of information from the “mesh control” category of tags described in previous subsections. Both categories of tags are displayed in the tag tree, but are edited using different dedicated user interface forms. Mesh control tags are used prior to the meshing operation, thermal tags are used after the meshing operation.

Thermal tags can be assigned to solids, 2D surfaces or faces of bodies, or edges. Any one object may have multiple tags assigned to it.

Thermal tags are created by selecting the desired item or items and then clicking the Thermal icon in the Edit Tags ribbon panel. The Thermal icon is only available if one or more items are selected. When the Thermal icon is clicked, the Thermal Editor window opens. A different form is available for each design geometry type (solid, surface and edge) as seen in Figure 4-23. The options available in the editor windows are determined by the items that have been selected. While multiple items can be edited at the same time, only one type can be edited at a time. If multiple types of items are selected, a selection filter (Figure 4-24) appears to allow selection of the single type that will be edited.

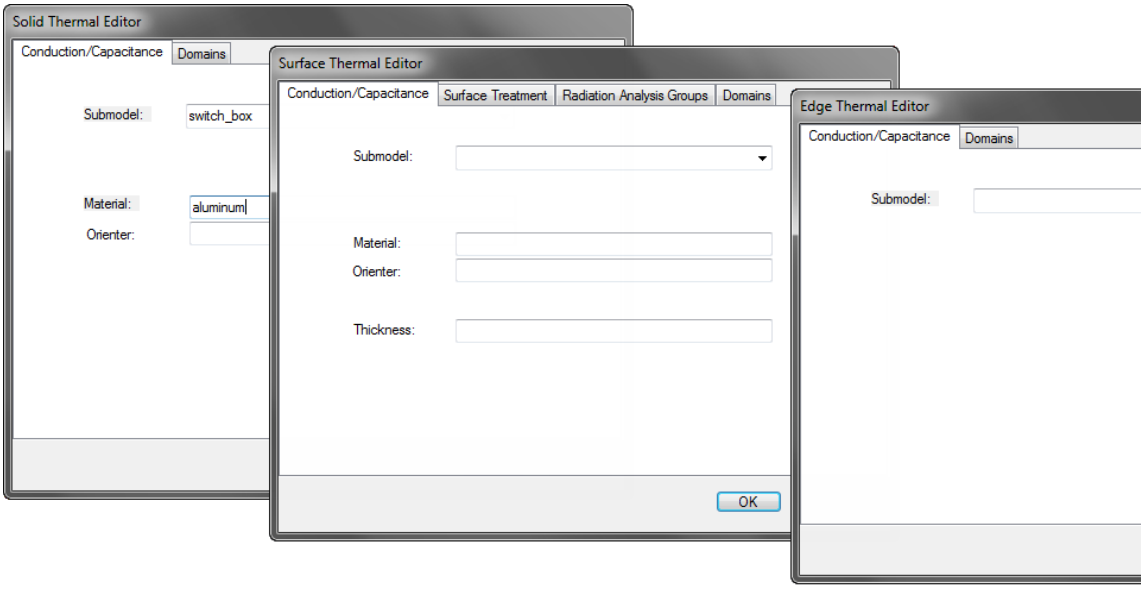


Figure 4-23 Thermal Editor windows for Solid, Surface, and Edge

For a solid, the thermal tag options are:

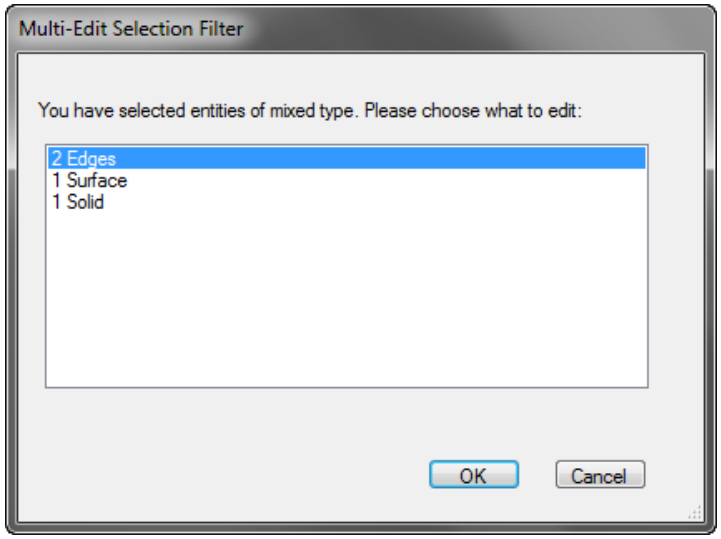


Figure 4-24 Multi-Edit Selection Filter for thermal tags

- submodel name (Section 4.4.1.1)
- material and orienter names (Section 4.4.1.2)
- domain names (Section 4.4.4)

For surfaces and faces of bodies, the tag options are:

- submodel name (Section 4.4.1.1)
- material and orienter names (Section 4.4.1.2)
- thickness (Section 4.4.1.3)
- optical property name (Section 4.4.2.1)
- insulation material name and thickness (Section 4.4.2.2)
- radiation analysis group names and active sides (Section 4.4.3)
- domain names (Section 4.4.4)

For edges, the tag options are:

- submodel name (Section 4.4.1.1)
- domain names (Section 4.4.4)

These options are described in the above-referenced subsections.

Some tags for faces, such as optical properties, insulation, radiation analysis groups, and domains, can be defined for specific sides of the face. The sides are positive (or top), and negative (or bottom).

The face of a solid is always positive away from the solid and negative into the solid. For free-standing surfaces that are not faces of a solid, the positive side can be determined by choosing the Measure ribbon and selecting the Normal icon in the Quality panel. In this mode, arrows are shown for the positive normal of any selected item. Optionally, colors can be used. See Figure 4-25 for an image of positive normal display.

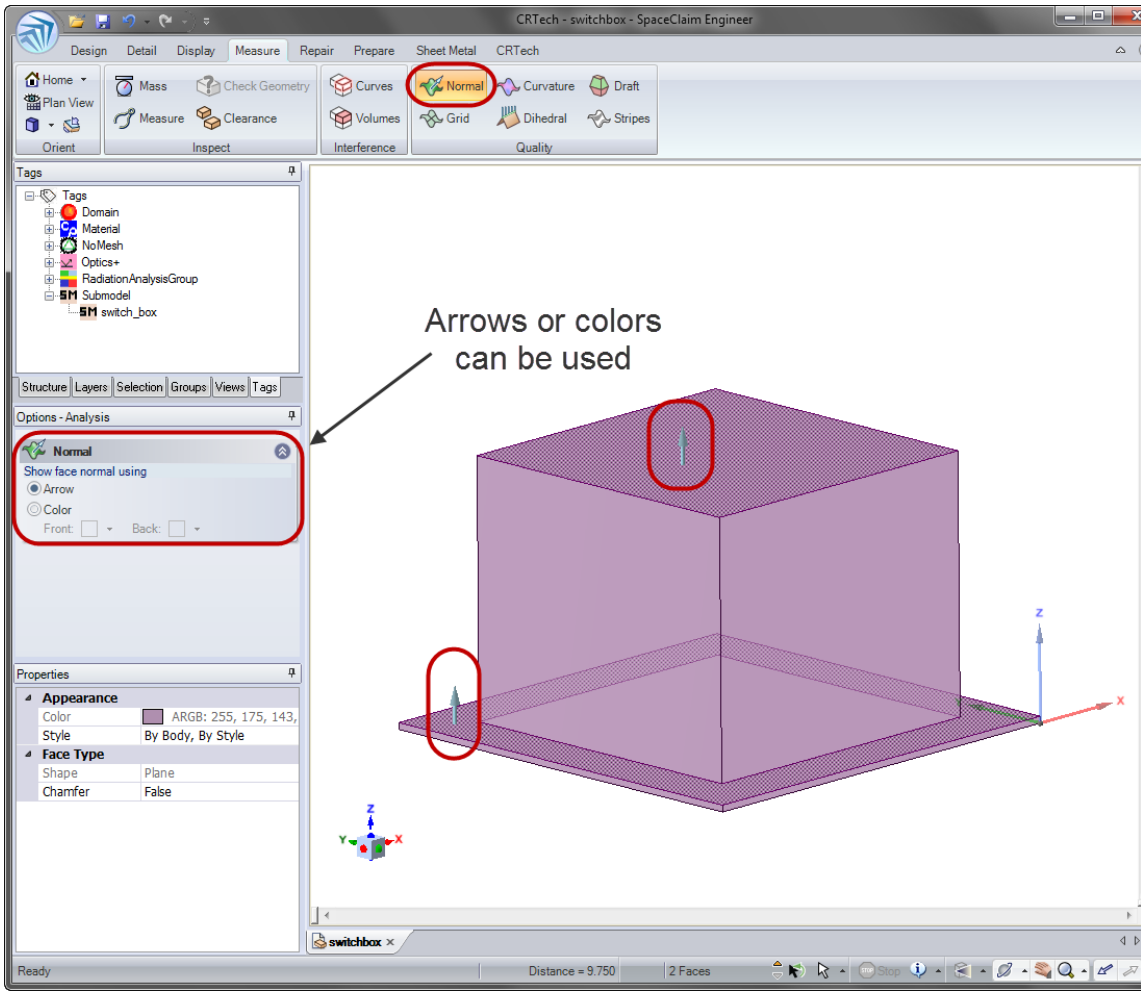


Figure 4-25 Showing positive surface normals using the Measure tools

All tag fields provide pull-down menus for accessing previously used values. In addition, some of the pull-downs will display values used by the associated Thermal Desktop drawing. Names of submodels, radiation analysis groups, material and optical properties, and material orienters are transferred from the Thermal Desktop drawing to the SpaceClaim document when a synchronize operation is performed in Thermal Desktop.

4.4.1 Conductance/Capacitance

The conductance/capacitance tab in the Thermal Editor window contains the submodel, material, and thickness tags as described in the next subsections. It also contains tags to specify a material orienter, and to specify that the geometry is to be used to align aniostropic materials.

4.4.1.1 Submodel

To specify the SINDA submodel for nodes and elements that will be generated, type the desired name of the submodel into the submodel field. If that submodel has already been used, it will be available as a drop-down choice to minimize typing. Submodel names that are currently in use by the Thermal Desktop model are listed in black. Submodel names used in the SpaceClaim document, but not yet defined in Thermal Desktop are listed in blue. After synchronization with Thermal Desktop, the submodels used in SpaceClaim will be defined in Thermal Desktop, and will appear black in the drop-down list.

All nodes and elements that are generated on the part of the tagged design geometry will be included in the submodel. If the submodel does not exist in the thermal model, it will be created. Submodel names are limited to 32 characters (the first character must be a letter) and may contain letters, numbers, and underscore ('_'). The names are case independent, meaning the name 'Box' is the same as 'BOX'.

If a submodel is not specified for an item, the submodel for the higher topological entity related to that item is used. For example, nodes and elements for a face are assigned to the submodel specified on the solid containing the face, unless a specific submodel is assigned to that face. As another example, nodes along an edge are assigned the submodel specified on the edge. If no submodel is assigned to the edge, the submodel for the face is used. If different submodels are specified on the faces that share an edge, the submodel from the face with the highest merge priority will be used. If no submodel is assigned to the face, the submodel assigned to the solid is used. If no submodel is assigned to the solid, the default submodel of "TD_MESH" will be used.

4.4.1.2 Material and Orienter

To specify the thermophysical material name for the 2D or 3D elements that will be generated, type the name of the material into the field. The material name will be assigned to elements after the meshing operation. The material name must be defined in the properties database of Thermal Desktop or used as an alias before a solution can be run. The drop-down list will show names of materials and aliases currently in use by SpaceClaim and also defined by Thermal Desktop. Material names that are used in SpaceClaim, but not defined in Thermal Desktop are listed in red. Any material name may be used in SpaceClaim, but it must exist in the database used by Thermal Desktop to enable an analysis to be performed.

If the material properties are anisotropic, a material orienter name can be added to reference an orienter that exists in the TD model. The drop down will show existing orienters in black, and undefined orienters in red.

Thermal Desktop contains a special type of orienter named “LOCAL”. If this name is used in the orienter field, then an internal coordinate system unique to each element will be used. Each element will contain a local coordinate system that can be aligned independently of all others. This local coordinate system is used to align anisotropic materials to specified surface normals and edges of the geometry.

Local element material orienter coordinate systems for elements can be generated by checking Use surface parametric coordinate system to define LOCAL material orienters for surfaces, and Use tangent to define +X for LOCAL material orienters for edges.

By default, the local material orienter coordinate system is aligned with the shape function coordinate system for each element. However, when a surface is used to align local material orienter coordinate systems, each solid element generated on the body to which the surface belongs is given a coordinate system that is parallel to the parametric coordinate system on the surface at the closest point to the surface.

If the Use surface parametric coordinate system to define LOCAL material orienters is checked, then that surface is called a “material orienter surface”. For each solid element generated on the body to which the material orienter surface belongs, a center point for that element is generated. This center point is tested against all material orienter surfaces that belong to the solid body. The closest point on the set of the material orienter surfaces for the body is used to generate a local material orienter coordinate system for the solid element. The surface that contains a point closest to the solid element center is used. A coordinate system is generated on the surface with the +Z normal to the surface, and the +X parallel to the parametric “u” direction (It may not be obvious how the u and v directions for SpaceClaim surfaces are oriented).

For example, suppose we have a thin curved solid, as shown in Figure 4-26, that is made out of an anisotropic material, with a through-thickness conductivity that is different from the in-plane conductivity. Also suppose that the temperature gradient through the thickness of the part is important, necessitating the need for a solid model rather than a mid-plane of 2D elements. Because the part is curved, each element will have a slightly different orientation.

We wish to align our material such that each solid element’s Z axis is perpendicular to the top surface. The two top surfaces are selected, edited, and the Use surface parametric coordinate system to define LOCAL material orienters is checked. The solid body must also be selected, and the name “LOCAL” used for the material orienter. As solid elements are generated, any surface on the solid body marked to orient the material is tested. The closest distance to each such surface is computed from the center of the solid element. The surface’s parametric coordinate system at the closest point is used to align the material for the solid element. This example results in the mesh shown in Figure 4-27. The field of orienters can be seen following the contour of the top surfaces of the part.

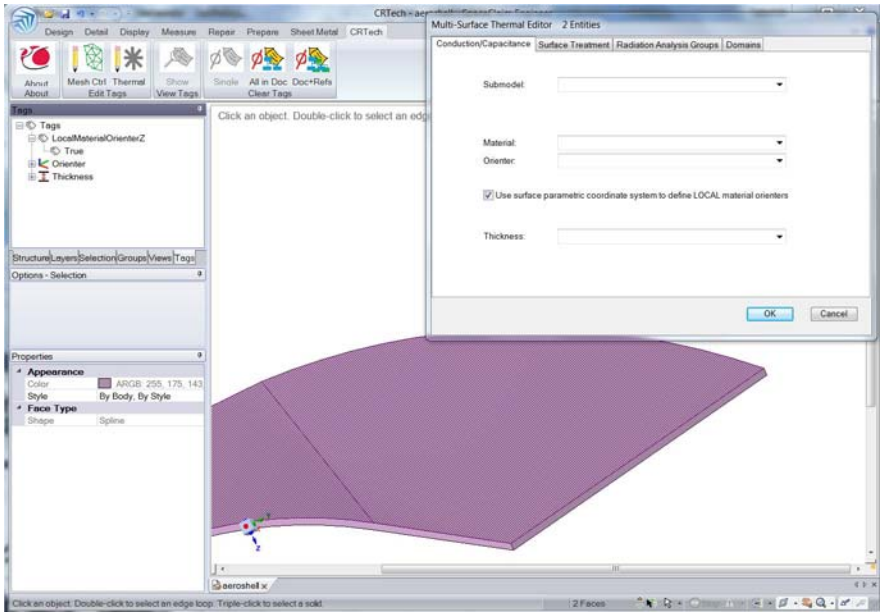


Figure 4-26 Surfaces selected as material orienter coordinate system generators

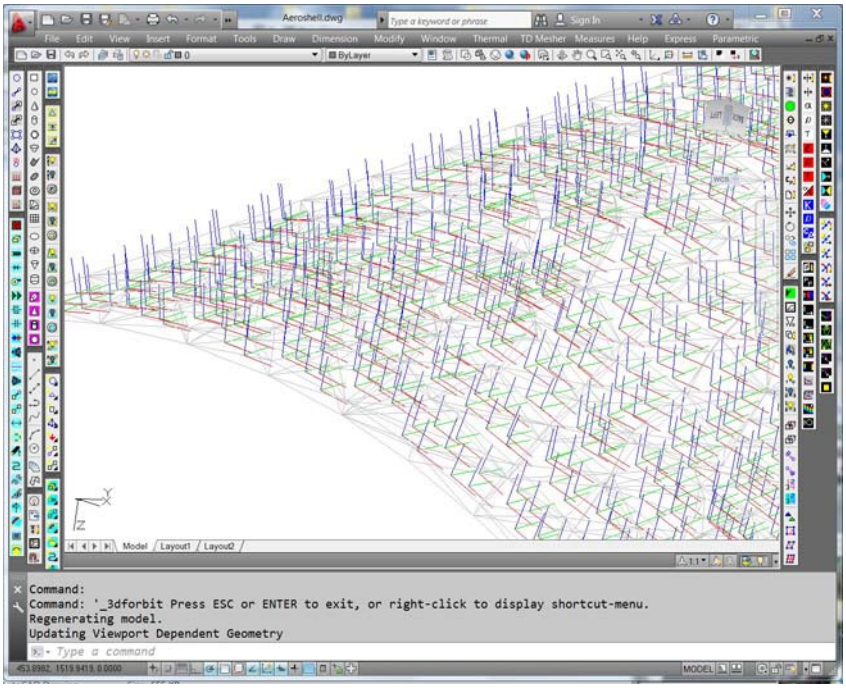


Figure 4-27 Solid mesh generated using surfaces to orient material

The placement of the material orienter coordinate systems can be verified in Thermal Desktop. Local material orienters will be drawn if the element specifies the material orienter name “LOCAL”, and the global visibility control for material orienters under Thermal Desktop’s user preferences is enabled. To hide the display of the coordinate systems, uncheck the global visibility option.

The “use for material orienter generation” tags are different from the other tags on the editing form. These tags do not control how elements are generated on the surface, but how solid elements are generated on the body to which the surface belongs. To generate local material orienters for a solid, use the orienter name “LOCAL” for the solid elements, and specify one or more surfaces on the solid to be used to orient the material.

The surface parametric coordinate system always aligns the +Z axis normal to the surface. However, the u and v directions may not be evident. If the material has different conductivities in the X and Y directions, it is necessary to also orient the coordinate system about the normal direction. To accomplish this, check the Use tangent to define +X for LOCAL material orienters for one or more edges on a surface that is being used as a material orienter. After the closest point on the surface to the solid element center is found, the closest point on the set of edges with the orienter option from the surface point is found. The tangent at this point is used to align the +X axis of the material orienter.

The procedure starts with the location of the center of the solid element. From the set of surfaces on the body that have the orienter option checked, the closest point on the closest surface is found. At that point, a coordinate system is generated with the +Z normal to the surface. The +X is generated parallel to the U direction on the surface. If the surface also contains edges with the orienter generation option checked, the closest point to the edges from the surface point is found. The tangent to the edge at the closest edge point is used as the +X axis. This coordinate system is then located at the center of the solid element.

Local material orienters can also be specified for surface elements. If the name “LOCAL” is used for the material orienter on a surface, a coordinate system normal to the element is generated and the +Z of the material orienter will be aligned with the element normal. If the surface also contains edges with the material orienter generation option, the tangent to the closest point will be used to align the +X axis. An example is shown in Figure 4-28, where the edge bisecting the top surface is used as a material orienter.

The generated surface elements are shown in Figure 4-29. It can be seen that the blue +Z axis is normal to the surface, and the red +X axis is parallel to the line forming the crease in the part. Local material coordinate systems are only displayed for surface elements when the element is generating conduction and capacitance data (which means the surface must be assigned a non-zero thickness in SCmesh), the element uses the material orienter named “LOCAL”, and the Thermal Desktop global visibility option is enabled for material orienters. The size of the axes is proportional to the size of the node display for both surface and solid element local material orienters

Surfaces and edges that have been selected to orient materials can be verified in the Tag Tree. All edges will be placed under the tag “LocalMaterialOrienterX”, and all surfaces will be placed under the tag “LocalMaterialOrienterZ”.

4.4.1.3 Thickness

A thickness may be assigned to surfaces and faces of solids. This thickness will be applied to the planar elements associated with those surfaces. If a design surface in SpaceClaim has a thickness property applied (as in a midsurface), that thickness will be applied

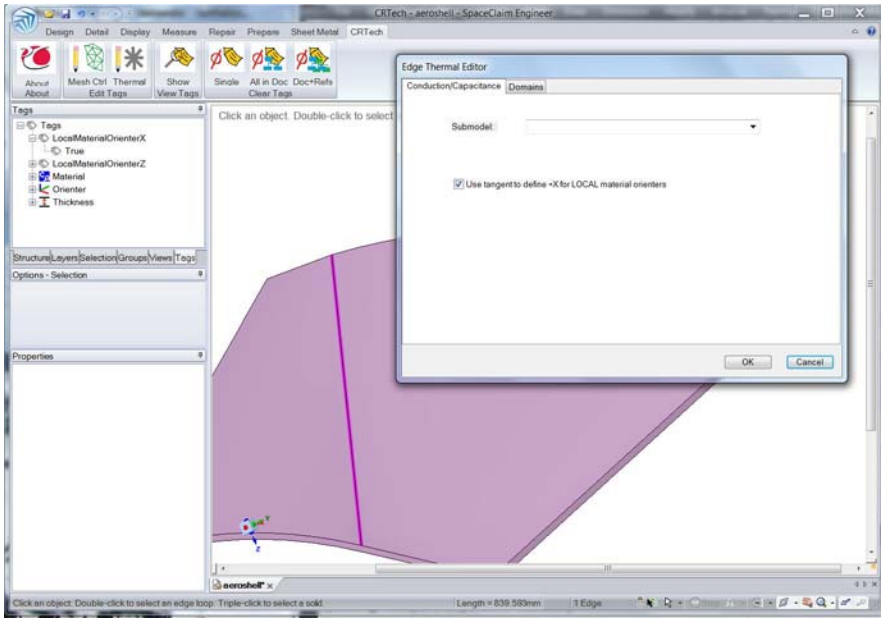


Figure 4-28 Edge used to align local material orienters for surface and solid elements

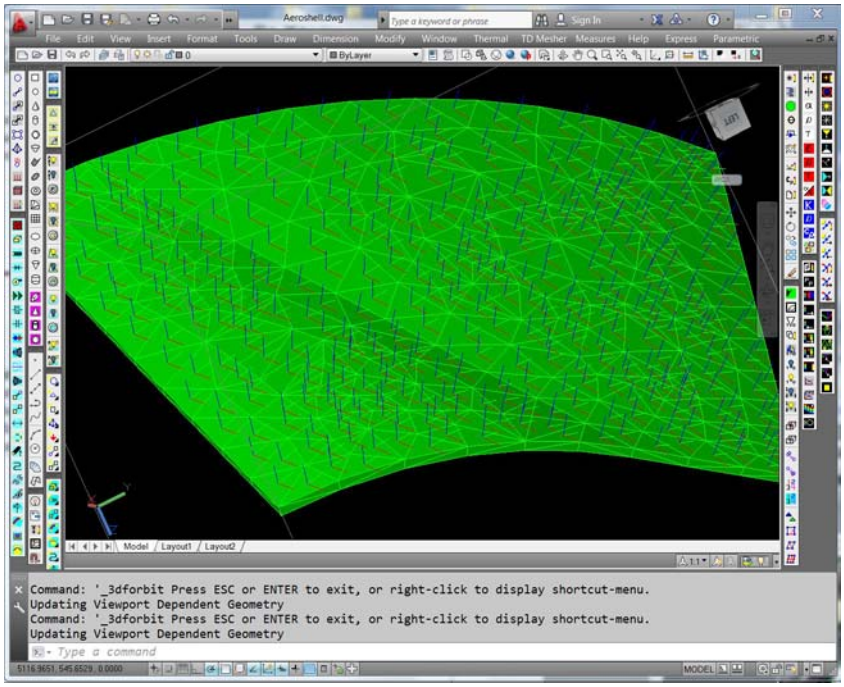

Command: '_3dforbit Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Updating Viewport Dependent Geometry
Command: '_3dforbit Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Updating Viewport Dependent Geometry
Type a command

Figure 4-29 Local material orienters generated for top surfaces that have the +Z axis normal to the surface, and the +X axis parallel to the crease in the part

to the surface elements unless a local tag is used to override the value. The drop down shows other thickness values used in the SpaceClaim document; no thickness information is imported from Thermal Desktop.

4.4.2 Surface Treatment

The surface treatment tab in the Thermal Editor window contains the optical property and insulation tags for the positive normal and negative normal sides of the surface (as defined above). This tab is available for editing surfaces or faces of solids.

> Note: At present, solid finite elements do not participate in radiation. If a face of a solid is assigned an optical property name that has a non-zero transparency, radiation will pass through the solid. This can be useful for modeling glass panels and lenses, but care should be taken to assign optical properties to both the positive and negative normal sides of all surfaces of a solid. Care should be taken to define the optical property in Thermal Desktop to include the proper index of refraction ratios, and to lump the total absorptivity through the solid as a surface property.

4.4.2.1 Optical Properties

Type the name of the optical property into the field for the appropriate side of the surface. In Thermal Desktop, the name must match a defined property or an alias before a solution can be run. The pull-down lists optical property names used by Thermal Desktop in black, and undefined names in red.

4.4.2.2 Insulation

For applied insulation, a material name and thickness may be specified for either side of the surface. This follows the same rules as the material and thickness tags (Section 4.4.1.2 and Section 4.4.1.3, respectively). Material orienters do not apply to insulation since the insulation conduction is only through the thickness. The pull-down menus function similarly to the material and thickness fields described earlier.

4.4.3 Radiation Analysis Groups

Radiation analysis groups may be assigned to surfaces or faces of solids. Surfaces are added to a radiation analysis group by clicking the Add button on the Radiation Analysis Groups tab of the Surface Thermal Editor. This button will cause the Radiation Analysis Group Name window to appear (Figure 4-30).

In this form, type the name of the analysis group (or select from the pull-down menu if that name has already been used in this document or is defined in Thermal Desktop) then choose the active sides. The definition of active sides can be found in the Thermal Desktop User's Manual. Any number of analysis groups may be assigned to a surface to allow for changing configurations. Radiation analysis group assignments may be edited or deleted by selecting the Edit or Delete button, respectively, on the Radiation Analysis Groups tab.

Radiation analysis group names are limited to letters, numbers, hyphen (-) and underscore ('_'). Spaces are not allowed. The names are not case sensitive.

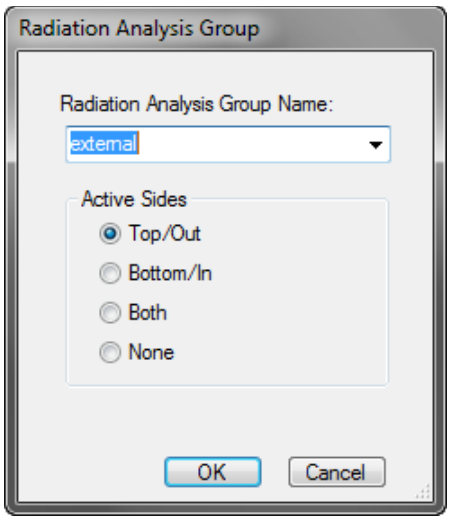


Figure 4-30 Radiation Analysis Group Name dialog

4.4.4 Domains

A domain is a special tag that can be applied to any edge, face, surface, or solid in the CAD model. Domains permit a variety of operations within Thermal Desktop by automatically generating Domain Tag Sets in the Thermal Desktop model. Unlike submodel name or property name tags, the corresponding Thermal Desktop usage of domains is not immediately obvious. Domains represent a powerful general-purpose mechanism for assigning "thermal meaning" to various subsets of the design geometry.

A typical definition of the word *domain* is "a territory over which rule or control is exercised." Thus, the domain of a given Thermal Desktop entity is the set of other Thermal Desktop entities upon which it operates. For example, the domain of a heat load is the set of nodes, surfaces, or solids to which the heat load is applied. A Thermal Desktop contactor has two domains: a *from* domain, and a *to* domain. Each domain is represented by a list of entities, with possibly additional identifiers to specify particular faces or edges.

The list of entities in a domain of a Thermal Desktop object is usually created by graphically selecting the items on the computer screen. These are specific or *direct* items in the domain of the Thermal Desktop object. It is also possible to *indirectly* specify what is in a domain of a Thermal Desktop object by using the name of a *Domain Tag Set.*

A Domain Tag Set (or just "domain set" for short) is a named collection of Thermal Desktop objects along with additional markers to indicate particular faces and edges if necessary. These collections are stored in the Thermal Desktop drawing. A Thermal Desktop object may use the name of a domain set in its list of domain items. This item is called an *indirect* (or *abstract*) domain item, since to the Thermal Desktop object, the actual objects that the Thermal Desktop object will operate on are hidden until a solution is launched. The domain of a Thermal Desktop object may consist of any combination of direct and indirect domain items.

Indirect domain items are resolved at the time an analysis is performed. The content of a domain set may be changed at any time without having to edit the Thermal Desktop objects that reference the domain set. Indirect domains permit Thermal Desktop objects to be wired together in a manner that allows easy modification without having to re-specify the basic topology of the connections.

To continue with the heat load example, instead of directly specifying particular nodes or surfaces on which to apply a heat load, a domain set name may be used instead. At the time that the domain set name is added to the heat load's domain, the domain set may be empty. It is merely a placeholder for other objects that will be resolved when an analysis is performed. As an example, say the name of a domain set used by a heat load is "friction_-surfaces." When a SINDA/FLUINT run is launched from Thermal Desktop, the heat load is applied to the *current* contents of the friction_surfaces domain set. The contents of friction_surfaces may be deleted or changed without having to edit the heat load, and the heat load itself is not lost if all the members of the friction_surfaces domain set happen to have been deleted.

The main purpose for applying domain *tags* in TD Direct is to automate the generation of domain *sets* in Thermal Desktop. Domain sets will be created that contain the results of the meshing operation on any design entity tagged with a domain name. For example, consider the faces of a solid part representing a brake pad. If the faces that contact the rotor are tagged with the domain name "friction," then two domain sets will be created in Thermal Desktop when the synchronize operation is executed. They will be named "friction_nodes" and "friction_surfaces" to indicate the type of members that each contains.

For each domain name tagged on engineering geometry in SpaceClaim, one or more domain sets will be created in Thermal Desktop. The name of the domain set in Thermal Desktop will be the name of the domain tag applied to the design geometry, plus a suffix appended to indicate the contents of the domain set. For domain names applied to a solid body, three domain sets will be created: one containing all the solid elements generated on the body that with "_SOLIDS" appended to the domain name, a second containing surface elements with "_SURFACES" appended, and a third containing the set of nodes in the solid (and on the surface of the solid) with "_NODES" appended. For surfaces, only node and surface element domain sets are created (see the *friction_nodes* and *friction _surfaces* example in the prior paragraph). Similarly, domains applied to edges also generate two domain sets: a set containing the nodes generated on the edge will be created, as well as a second set that consists of the element edges with "_EDGES" appended to the domain name.

This scheme is summarized by the following generated domain set names, assuming that three domain tags were applied to a solid, a surface, and an edge, and that these domains were name MYSOLID, MYSURFACE, and MYEDGE, respectively:

MYSOLID_SOLIDS
MYSOLID_SURFACES
MYSOLID_NODES

MYSURFACE_SURFACES
MYSURFACE_NODES

MYEDGE_EDGES
MYEDGE_NODES

In summary, a domain tag applied to one or more CAD entities causes domain sets to be generated in Thermal Desktop, which then may be used as indirect domain items in Thermal Desktop entities that operate on a collection of other Thermal Desktop entities. Restating the heat load example, once the indirect domain item “friction_surfaces” is added to the heat load, any remeshing operation or redefinition of the CAD entities included in the domain “friction” automatically applies the heat load to the new set of finite elements that are generated from that CAD geometry. TD Direct will automatically create new domain sets in Thermal Desktop, which are automatically applied in the domains of the Thermal Desktop objects that use them.

Domain names are also used within Thermal Desktop through Mesh Editor Action Scripts (Section 2.3.3). The usage of these scripts will become more clear after this section has been read.

On the Domains tab of the Thermal Editor, selecting the Add button will open the Domain dialog (Figure 4-31) to allow a domain name to be entered or chosen for the selected geometry. For solids and edges, just the domain name is specified or selected from the pull-down menu. For surfaces, the domain name is specified or selected *and* the desired side for this domain is chosen: positive normal, negative normal, or both. The pull-down

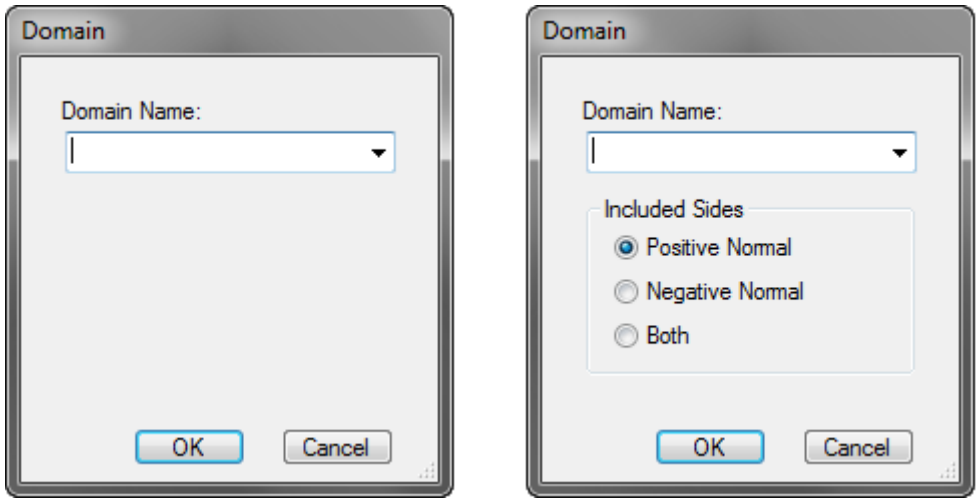


Figure 4-31 Domains dialog for solids and edges (left) and surfaces (right)

lists previously used domain names in the SpaceClaim model. No synchronization with Thermal Desktop is performed.

Care should be taken to make sure that domain names are unique to each SpaceClaim model imported in Thermal Desktop. A SpaceClaim model may have one or more dependent documents; only the top level document is referenced by the Thermal Desktop SpaceClaim importer. Domains can span the set of SpaceClaim documents used for an assembly model.

During a synchronization, the contents of domain tag sets are replaced. If one or more importers reference SpaceClaim documents that use the same domain name, the last importer to synchronize will replace the results of the other importers.

The domain names are used to automatically generate domain sets whose name is based on the domain name plus an appended character string. Since domain sets have a name length limit of 31 characters in older versions of AutoCAD, **surface domain names should be limited to 22 characters** to allow "_SURFACES" to be appended; **solid and edge domain names should be limited to 25 characters** to allow for "_NODES" (and other equally long strings) to be appended.

Allowable characters in domain names are letters (A-Z), numbers (0-9), hyphen ('-'), and underscore ('_'). The names are not case dependent since they will all be converted to uppercase.

In summary, domains on engineering geometry in TD Direct create domain tag sets in Thermal Desktop, which can be used in the domain of Thermal Desktop entity, such as a heatload or contactor. It is a way that entities in Thermal Desktop can virtually use higher level CAD geometry in their usage list. Using the name of a domain tag set in a Thermal Desktop's list of entities is like specifying the CAD geometry itself.

4.4.4.1 Generated Domain Tag Sets

When an engineering geometry item is tagged with a domain name, the nodes, elements, and element edges that are generated on those engineering geometry entities are placed in automatically generated domain tag sets in Thermal Desktop. The domain tag sets will be named *domainName_type*, where *domainName* is the domain name assigned in TD Direct and *type* is the domain tag set type and a descriptor of the contents of the domain tag set. For example, selecting a face and tagging it with the HEAT_LOAD_1 domain will generate two domain sets in Thermal Desktop: HEAT_LOAD_1_NODES and HEAT_LOAD_1_-SURFACES.

Additional domain sets are automatically generated when surfaces or edges are matched. When the matched option is used in the default mesh controls, surfaces and edges that are in contact are imprinted on each other. Separate, but identical matched meshes are produced in the contact region. For every domain on a matched entity, additional domain sets will be created that contain only the subset of the nodes, elements, or edges in the matched region.

The text "MATCHED_FACE_<n>" or "MATCHED_EDGE_<n>" will be appended after the domain name. For example, if a box is on a plate, and the plate is assigned the domain name "PLATE," then domain tag sets will be created that are named "PLATE_-MATCHED_FACE_1_SURFACES" and "PLATE_MATCHED_FACE_1_NODES." The numerical index indicates which region is matched with other domain names. In this example, if more than one box was on the plate, each matched domain would be identified by a unique integer. If a domain is on the box, additional domain sets will be generated using the "MATCHED_FACE_<n>" text string. The integer identifier will be the same for the matched faces or edges on each of the domains that are in contact.

This allows specific actions to be applied to only the matched regions. For example, a contactor may specify the *to* and *from* regions that only contain the precise areas or edges that exactly contact. Since the meshes match up identically, a contact integration interval of 1 may be used to speed contactor calculations. Matching domain tag sets can be verified in Thermal Desktop by using the Model Browser, and listing by domain tag sets. Use the "only" (and other) display options to verify matching domain tag sets.

4.4.5 Measurement Symbols

With more complex thermal models, it may be necessary to have additional information about the geometry available in Thermal Desktop. The location of a vertex, the length of an edge, the area of a surface, or the volume of a solid can be transferred from TD Direct to a Symbol in Thermal Desktop by using the Measurement Symbols (see Thermal Desktop User's Manual for more information on Symbols).

When a vertex, edge, surface, or solid is selected, clicking the Measurement Symbols button will bring up a window for the associated type of data (position, length, area, or volume). The user can create a Symbol name and comment. When the models are synchronized, the Symbol will be entered in Thermal Desktop with the correct value. The values will be updated with each synchronization.

In the case of a vertex, three Symbols will be created in Thermal Desktop. These Symbols will use the name defined in TD Direct with _X, _Y, and _Z appended to them to indicate the coordinates.

Symbols will be in the base units of **meters** to prevent issues with user logic should the model units be changed.

4.4.6 Grip Manipulators

Grip Manipulators (see Thermal Desktop User's Manual) can be created on vertices. These Grip Manipulators are given names and optional comments in TD Direct, and will appear in Thermal Desktop after a synchronization. Additionally, new Symbols with _X, _Y, and _Z appended to the name of the Grip Manipulator will be created and entered into the position definition. These new Symbols will be in units of **meters**, and the implementation on the position tab of the Grip Manipulator will be in the current drawing units.

4.5 Assemblies and Multiple Instances

A powerful feature of TD Direct is the ability to analyze complex assemblies. Where components of the assembly touch, the mesh may be contiguous across the interface, representing a single fused part. Alternatively, each side of an interface may be meshed separately but identically across the interface ("matched"). The matched meshes can be used with a Thermal Desktop contactor to create a thermal connection (see Section 4.3.1.5 "Mesh Type for Contacting Geometry").

It is assumed that the user is familiar with the basic assembly concepts of SpaceClaim. If not, a good overview is presented at http://www.spaceclaim.com/en/Support/Tutorials/Modules/SpaceClaim_Assemblies_Tutorials.aspx. Please review if necessary.

Tags should be applied at the lowest level possible of an assembly structure. For example, you might construct a document that has representation of a turbine blade (say, in the file blade.scdoc), then insert that component multiple times in another document to create a turbine (turbine.scdoc). You can select all the blades in the assembly model (turbine.scdoc) and assign a material, or you could select the single blade in the blade.scdoc and apply the material there. Applying information to the single blade is easier and less prone to mistakes when making changes.

When blade.scdoc is inserted into turbine.scdoc, all tags defined in blade.scdoc are "inherited" in turbine.scdoc. Each blade in turbine.scdoc will have the properties that were assigned in blade.scdoc. If changes are desired, editing the part in blade.scdoc guarantees that all blades receive the changes.

As a general rule, all information that will be common to all instances of a component should be placed at the lowest level of that component. If particular instances require different properties, they can be applied at any higher level of the assembly. In turbine.scdoc, any particular blade can be edited and the properties changed for that blade. Additional tags in this case exist on the component in the file turbine.scdoc, as well as on the original part in blade.scdoc. The process flow is illustrated in Figure 4-32.

As a component is inserted into another SpaceClaim document, all tags defined in the component will be inserted (*inherited*) as well. In addition, a tag applied in the containing (higher level) document may overwrite any tag applied in an inserted (lower level) document. This behavior is maintained for all levels of the assembly. If turbine.doc is inserted as a component into yet another document, say turbocharger.scdoc, then all tags applied in blade.scdoc and turbine.scdoc are inherited by turbocharger.scdoc. Inheritance and overwriting is maintained throughout the assembly structure.

The Tag Tree shows the net result of all inheritance and overwriting operations for the currently active document. It shows the tags that will be used for the meshing operation and the assignment of thermal information to the mesh in Thermal Desktop. The Tag Tree will not show where exactly in the assembly sequence a particular tag was applied, just the net result. If you wish to trace the application of a particular tag, use the Show icon. The top pane will show tags applied to items in the currently active document, the lower pane will show the net result at this level of the assembly. If the tag you are investigating is not listed in the "local" pane, then it was inherited from a lower level of the assembly structure. Open the document containing that component and repeat the process there until you find it listed in the local pane. Any tags listed in the local pane have been applied in the currently active document.

The preceding example shows how a single part can be instantiated multiple times in a SpaceClaim assembly. The top level assembly in this case is what is imported into Thermal Desktop. If turbine.scdoc is imported into Thermal Desktop, each blade will have a unique

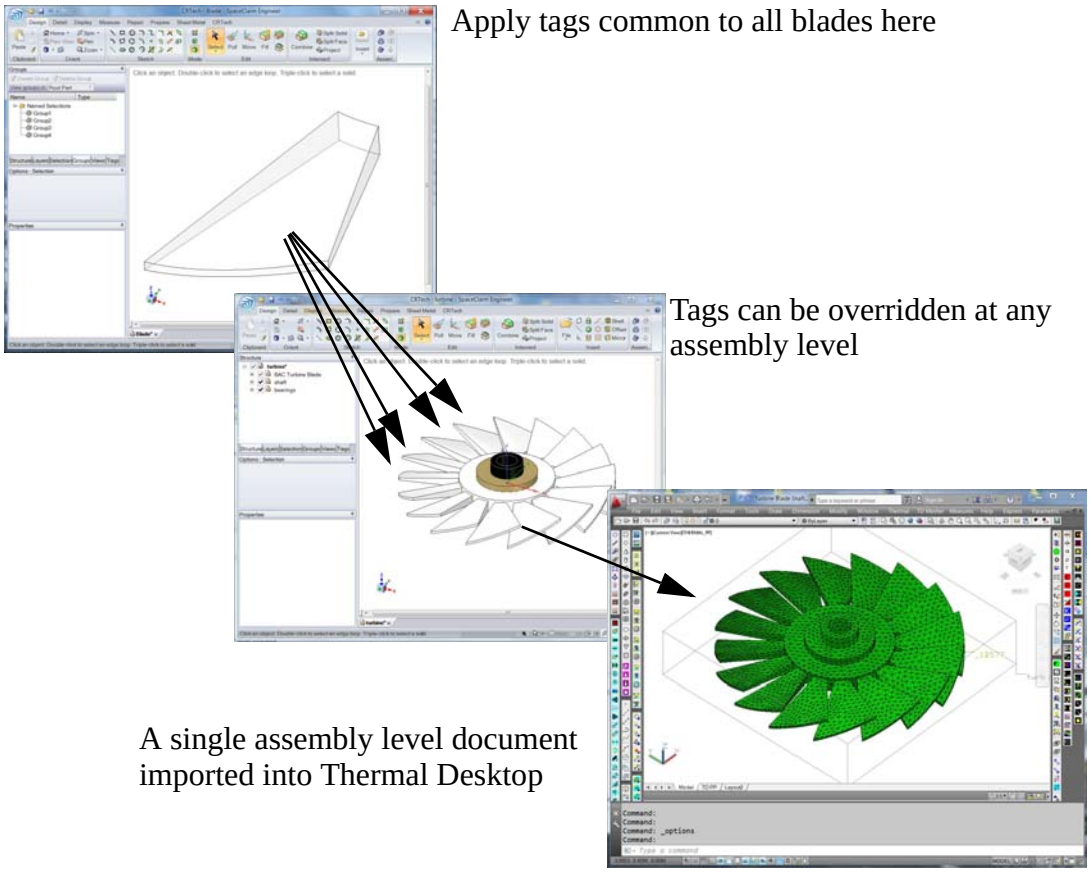

Apply tags common to all blades here
Tags can be overridden at any assembly level
A single assembly level document imported into Thermal Desktop

Figure 4-32 Multiple instances of a part created in a SpaceClaim assembly before importing into Thermal Desktop thermal model

node ID, since all blades were meshed together as an assembly. But what if you have a model that needs to be included multiple times in Thermal Desktop, but there is no “parent” assembly that can also be constructed in SpaceClaim?

Consider for example a model of thruster on a spacecraft. Say a detailed geometric model has been constructed to investigate the possibility of propellant lines freezing, and to determine the peak temperatures during operation. This thruster design may be used in multiple locations on the spacecraft. For efficiency, a simplified finite difference model of the spacecraft bus has been constructed in Thermal Desktop. It is desired to include the detailed thruster multiple times in the thermal model at various locations on the spacecraft bus.

A SpaceClaim model of the bus does not exist, therefore it would be very difficult to correctly position each thruster. With some effort, such a top level assembly could be constructed by exporting information from the TD finite difference model, but let’s explore an easier method.

The model of the thruster, say in thruster.scdoc, could be imported multiple times into the Thermal Desktop drawing. Each TD SpaceClaim Importer would point to the same SpaceClaim document. Each importer can be positioned as desired in the system level model created in Thermal Desktop. However, without further action, each included instance of the

thruster is identical. Each instance will be created with the same submodel and node ID information: all thrusters would have the same temperatures, regardless of their position, since mathematically they *are* the same thruster, despite appearances. While this method is useful for reduced model size by exploiting symmetry, in this case it is inappropriate because each thruster is not exposed to the same environment: each should have unique temperatures. Therefore, each instance as a minimum must be created with unique submodel and node IDs.

An editor action script (Section 2.3.3) may come to mind as a possible solution, but the domains on which the scripts act would not be unique. In other words, since each instance would be defined with the same domain, and domain tag sets would be overwritten. Each instance could be resequenced manually, but this process would have to be repeated for each design change and synchronization, negating the automatic associativity between the design and thermal models, and opening up the possibility for an incorrect thermal model if this resequencing step is forgotten.

To solve this problem, simply take the base thruster model, and use it to create a set of a unique assembly documents, one for each actual instance of the thruster. All such documents can be accessed within the same SpaceClaim session, as shown by the tabs at the bottom of the display window in Figure 4-33. Simply open a new design (e.g., "zenith_thruster.scdoc"), then use the Insert>File command to insert the original thruster.scdoc document, which remains intact and independent. Then, in the Thermal Desktop model, create SCImporters for each assembly document (each instance of the thruster). This avoids the problems associated with importing the same document multiple times, as previously described, yet centralization of design data has not been lost, as will be described below.

You might create a SpaceClaim document called nadir_thruster.scdoc, and insert thruster.scdoc. Then perhaps another document called minusX_thruster.scdoc and insert thruster.scdoc into that assembly as well. Now in Thermal Desktop, import nadir_thruster.scdoc with one SCImporter, and import minusX_thruster.scdoc with another, and so on. Uniqueness in the Thermal Desktop model will be effected by the application of additional tags in the top level assembly SpaceClaim documents.

For information that is common to all thrusters (most likely material and optical properties, thicknesses, and radiation analysis groups), apply tags in thruster.scdoc. For information that is different for each instance (for example, regions for contact with the spacecraft bus), apply the tags in each separate assembly document. For example, in nadir_thruster.scdoc, assign the submodel "NADIR_THRUSTER", and in minusX_thruster.scdoc, assign the submodel "MX_THRUSTER". Now each instance in Thermal Desktop will have nodes in unique submodels. The assembly documents contain unique tags (such as submodels and domains) plus the common/centralized thruster design (and any tags common to all instances of the thruster, such as materials) that is shared with all other assembly documents.

Likewise for domains intended for contactors, use unique names at the higher assembly level. In nadir_thruster.scdoc, apply the domain "nader_mounting_pad" to the appropriate surfaces. In minusX_thruster.scdoc, use something like "mx_mounting_pad". Unique domain tag sets will be created in the Thermal Desktop model for each thruster. Figure 4-34 shows the overall process for creating multiple copies of the same part in Thermal Desktop.

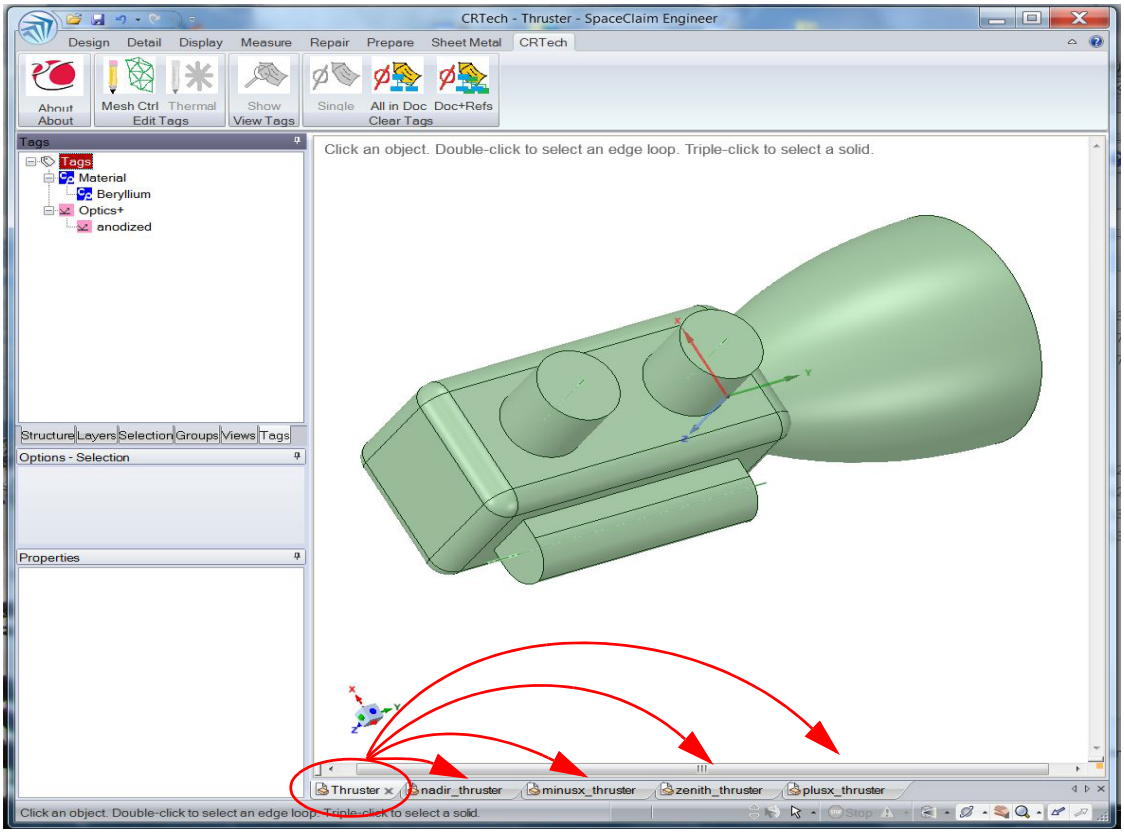


Figure 4-33 Base component used for multiple instances in Thermal Desktop model

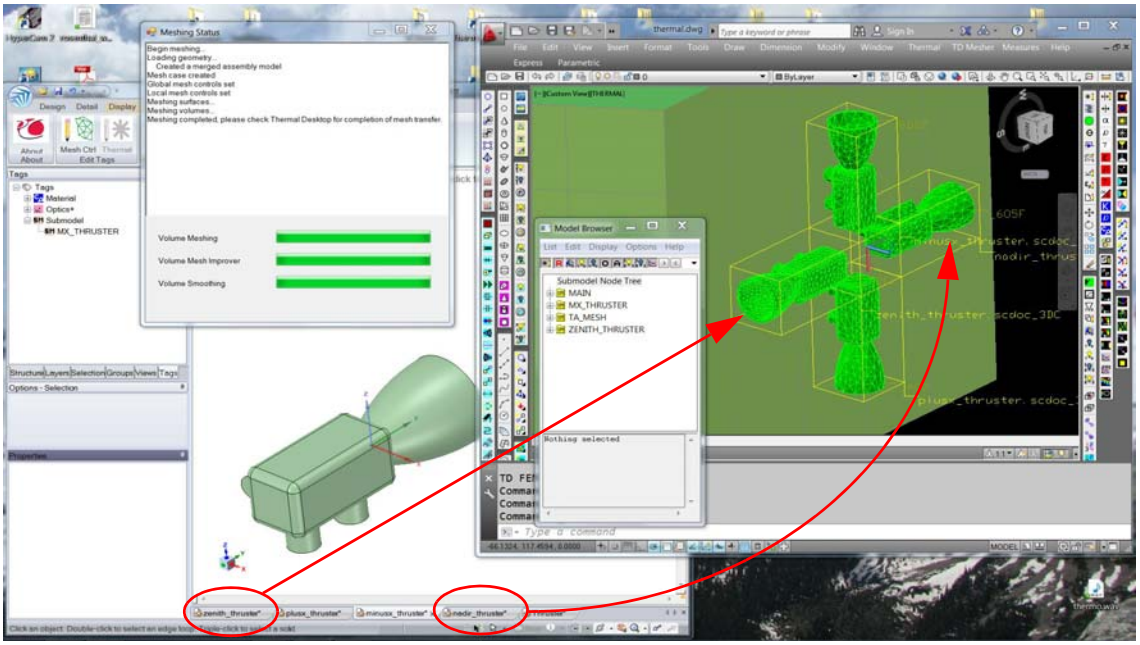


Figure 4-34 Uniquely node numbered instances of a base component imported into a Thermal Desktop model

The figure shows that a submodel tag has been added to minusX_thruster.scdoc to assign the submodel MX_THRUSTER to this instance of the thruster. The model browser in Thermal Desktop shows the submodels for each instance of the thruster, and the graphics area shows the four unique thrusters. In this example, it is also shown that different mesh controls can be used for each instance. The zenith thruster is meshed more finely, perhaps to answer a question requiring a detailed investigation, but without the burden of having to mesh all thrusters with such detail.

Organization in this manner allows unique thermal instances to be created in the Thermal Desktop model and positioned arbitrarily. Commonality is still maintained in thruster.scdoc. Any changes to this document, whether dimensions or thermal properties, will propagate automatically into each instance in the Thermal Desktop model when the Thermal>TD Direct>Synchronize Links command is selected.

Two styles of assembly modeling are possible. The first style consists of creating a single top level SpaceClaim document that pulls in the desired components from other supporting SpaceClaim documents. This single top level SpaceClaim document is imported into Thermal Desktop. The other style consists of making unique top level SpaceClaim documents for each desired instance of a component, and importing each separate document into Thermal Desktop. Each separate top level SpaceClaim document should have tags to differentiate the instances from one another. Either or both styles can be employed with the same Thermal Desktop model as appropriate.

4.6 Troubleshooting Unsuccessful Meshing

The meshing algorithms in TD Direct are robust and contain many unique features to handle difficult geometry. However, it may not be able to generate meshes for some combinations of mesh controls and/or geometry with defects.

When a mesh operation is unsuccessful, a message box will appear in Thermal Desktop indicating there was a problem with generating a mesh. Close this dialog and bring the TD Direct process to the front if necessary, by clicking on the TD Direct main window, or the TD Direct/SpaceClaim icon in the task bar. A pop-up message, such as shown in Figure 4-35, will appear.

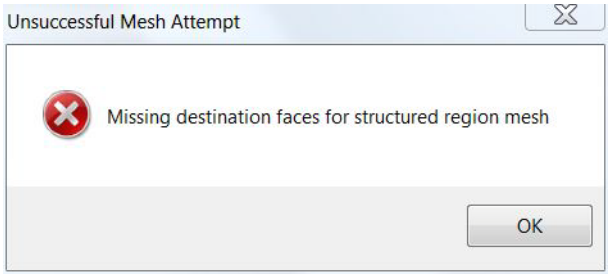


Figure 4-35 Unsuccessful Mesh Attempt message box

If there are engineering geometry entities associated with the message, they will be colored red, and highlighted in the graphics area. If the problem is associated with a small edge, all of the vertices of that small edge will be highlighted with a halo around them to make them more visible. The highlighting will be cleared when another SpaceClaim tool is selected, or the <ESC> key is pressed.

Here is a list of actions to try to resolve an unsuccessful meshing operation:

- Be sure you have generated geometry that is appropriate for analysis. Attempting to mesh detailed design geometry as-is will most likely fail.
- Zoom-in on corners and edges and verify the geometry is “water tight” where expected.
- Repair the geometry using the Repair tools in SpaceClaim. These tools repair small gaps, small faces, split edges and other geometry problems that can adversely affect mesh generation. Engineering geometry should pass all checks available in SpaceClaim before attempting to mesh. (See Section 3.3.1 "Repair".)
- Highlight the top level item in the Structure Tree, and select the Check Geometry button on the Measure tab. Correct any defects reported by this tool.
- Check for interferences and clearances using the tools on the Measure tab.
- Quadrilaterals can be problematic when they do not naturally fit the geometry well. Use quad dominant or consider using structured meshing.
- Try different mesh resolutions (sizes and curvature refinement). Meshes may fail if too coarse compared to underlying geometric complexity. Consider simplifying the geometry, or use a finer mesh.
- Small triangles are more likely to succeed than coarse quadrilaterals.
- Make sure you have not exceeded the maximum allowable number of nodes.
- Do not try to suppress “large” features using the suppress small features parameter. Features to be suppressed must be smaller than the intended mesh size. In general, it is better to remove features from design geometry using the tools in SpaceClaim.
- Check for limitations of structured meshing. (See Section 4.3.6.1 "Swept Mesh Limitations".)
- If importing from another CAD system, try to use the native representation. Translations to STEP or IGES often introduce defects.
- For very complex imported CAD geometry, consider simplification by creating a new representation using key features from the imported model, rather than attempting to directly transform the imported geometry. (See Section 3.3.3 "Simplification".)

If the above steps do not fix a failed meshing operation, please contact CRTech for assistance.

5 Tutorials

These tutorials for TD Direct assume the user has some basic familiarity with Thermal Desktop. Other tutorials in the Thermal Desktop Manual should be explored prior to attempting TD Direct if the user has no experience with Thermal Desktop.

Additionally, basic operations of SpaceClaim should be learned through the videos available on the SpaceClaim website.

http://www.spaceclaim.com

Tutorials may be found in the *\Tutorials\TD Direct* subdirectory, which is located in the Thermal Desktop installation directory (usually *C:\Program Files\Cullimore and Ring\Thermal Desktop*). The Tutorials subdirectory includes additional subdirectories, one for each of the tutorials covered in this manual. It is recommended that the Tutorials directory be copied to the user's own working area before beginning the tutorials. This ensures a copy of the original tutorial files will be available for use by other users at a later time.

5.1 Basic Mesh Tutorial

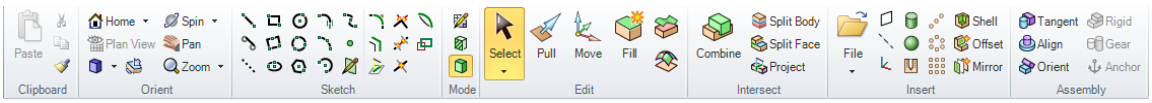
This tutorial starts with the design geometry of a fin used to cool a diode using SpaceClaim and ultimately creates and runs a thermal model of the system in Thermal Desktop. The steps will include using the Fill tool to simplify the drawing, meshing the parts, using Domain Tags, working with assembly interfaces in the mesh, utilizing local mesh controls, and implementing Driving Dimensions from Thermal Desktop. Completed files are provided at various stages to allow the user to compare and check progress.

1. Open ***HeatSink_1.scdoc*** in *\Tutorials\TD Direct\Fin_Diode*.

   This is a simple block with several fins that can be used to enhance convection. It will be used throughout the tutorial.

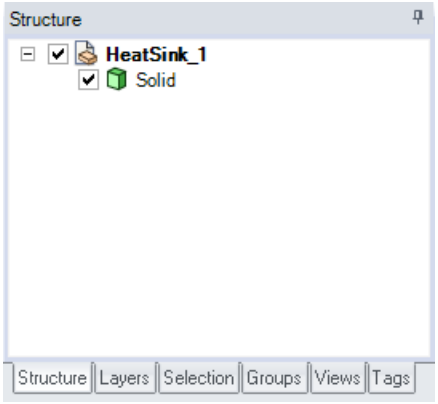
   Let's examine the user interface. SpaceClaim uses a Ribbon Tool Bar interface, and the first tab of the ribbon is Design. This is where you will do most of your operations to create parts. All of the TD Direct-specific tools can be found on the Thermal tab.

   Design Tab



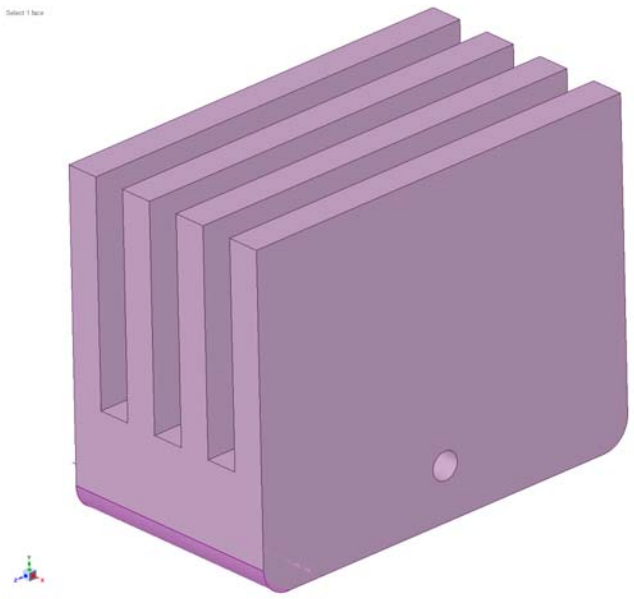
Thermal Tab

On the left side of the interface is the Structure Panel, which has tabbed windows for Structure, Layers, Selection, Groups, Views, and Tags. To start, select the Structure tab so that you can see the design body (Solid) defined in the component HeatSink_1.

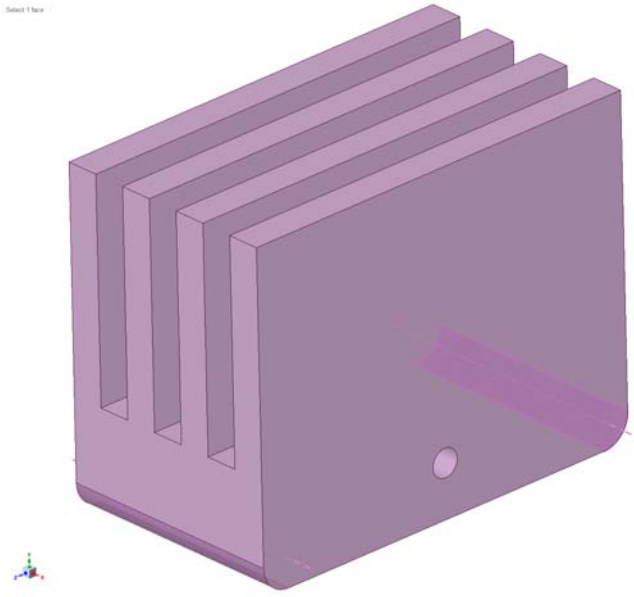


Spend a few minutes exploring the different tabs. Many of these will be used in the tutorial.

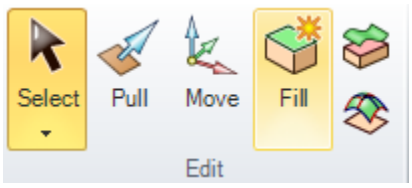
2. There is unnecessary detail in the geometry that should be removed before attempting to make a thermal mesh of the component. In your own models it will be up to you to determine the appropriate level of detail. Select the round on the bottom left of the fin by moving the cursor over the round and clicking once.



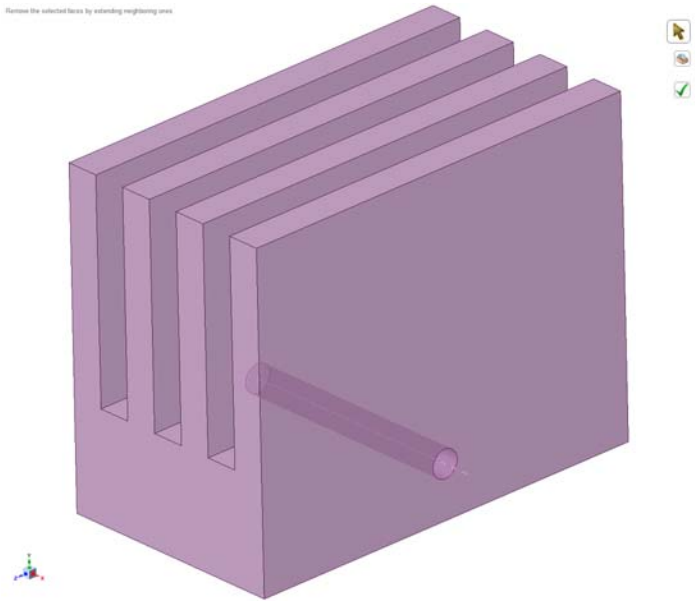
3. Move the cursor to where the second round is even though it is hidden on the other side of the solid. Use the center scroll wheel until the round is highlighted. Select it while holding **Ctrl** so both rounds are selected at once.



4. Go to the Design tab and select Fill. Both rounds will be squared off. In this case you are selecting prior to starting the operation.

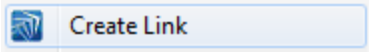


5. A bigger concern is the hole through the heat sink. Press **Esc** to ensure nothing is selected, and then select Fill again. Move the cursor to the hole so it is highlighted and select it with a single click. Click on the green checkmark on the right side of the display, which accepts the inputs for the Fill command. This is selecting after starting the operation. SpaceClaim's website has many video tutorials about simplifying geometry for analysis, including a more powerful tool for removing complex rounds.

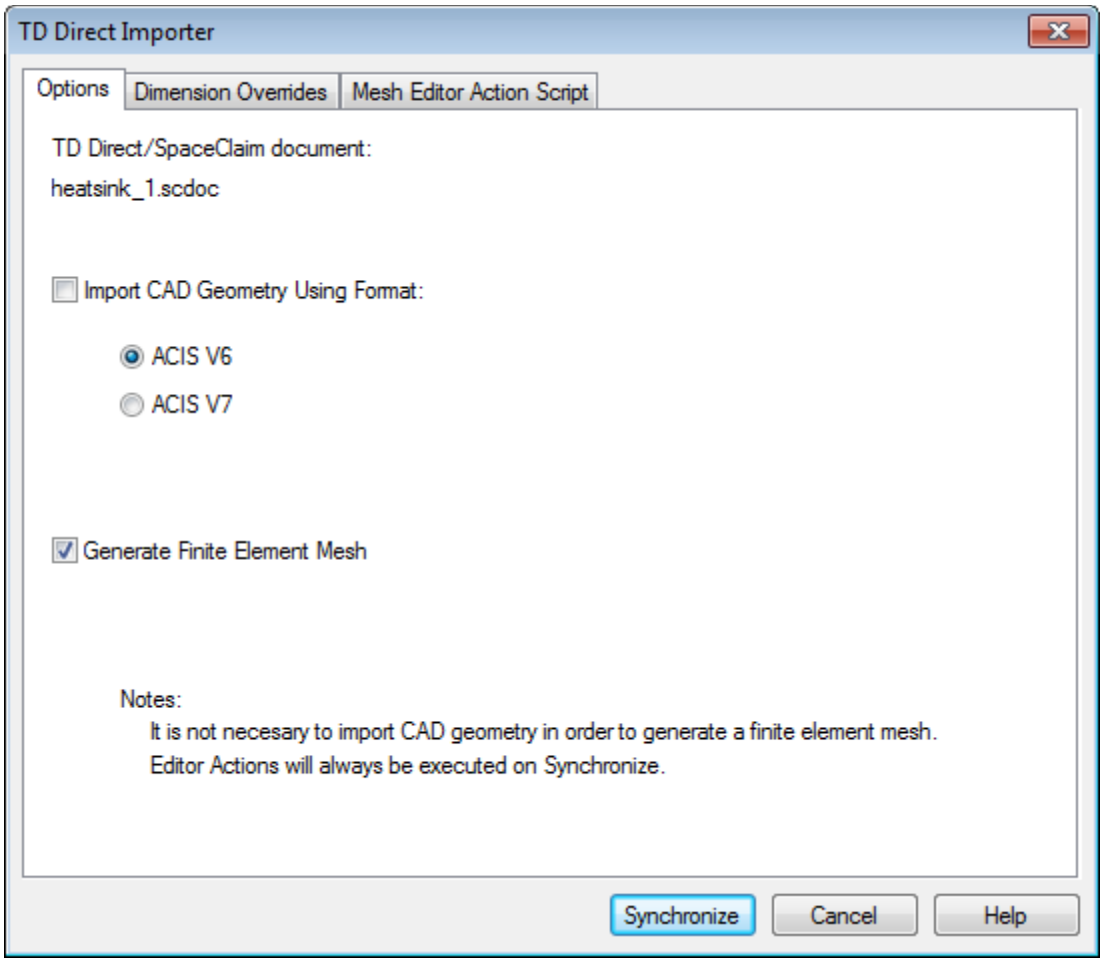


6. Open the Thermal Desktop file ***HeatSink_1.dwg***. This model has a single boundary node defined to represent air. All thermal analysis will be done in Thermal Desktop.

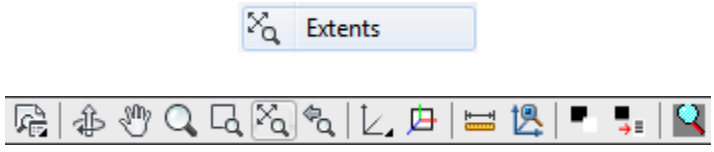
7. To create a link between the Thermal Desktop model and the TD Direct document, use the Thermal Desktop pull-down menu to select *Thermal > TD Direct > Create Link*. Select the working TD Direct document (HeatSink_1.scdoc).



8. Deselect the box labeled Import CAD Geometry Using Format and select the box labeled Generate Finite Element Mesh. Click on Synchronize to establish the link between Thermal Desktop and TD Direct. It may take a few minutes for the mesh to be generated.



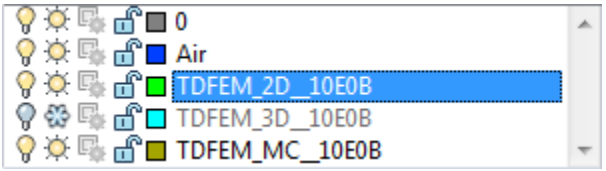
9. Use Zoom Extents as necessary to bring the mesh into view (*View > Zoom > Extents* or using the tool button shortcut).



The link to the TD Direct document is represented as an object in Thermal Desktop, and may be selected and edited as any other Thermal Desktop object. It will be created on the currently active layer. Graphically it shown as a bounding box around the mesh, with a label to the right indicating the name of the TD Direct document with a special identify appended.

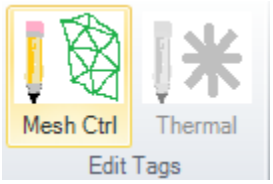
10. Change View to Thermal Shaded using the tool button shortcut.

11. Observe the layers. Three layers are created: 2D, 3D, and the Mesh Controller. By default, the layer of the surfaces is turned on, and the layer of the actual solids are turned off and frozen. The bounding box and the text label will be placed in the layer that was active when the link was created (in this case, Layer 0). More information regarding layer usage with TD Direct can be found in a video on the CRTech website.



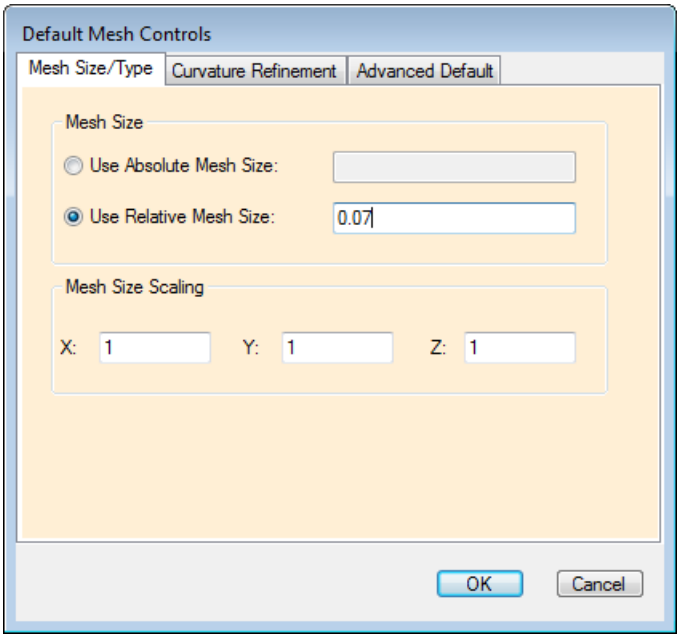
12. Return to TD Direct. If you are switching back and forth frequently, as you will be in this tutorial, you may want to spend a few moments thinking about how to set up your windows. If you use two monitors, you may want to put them on different screens. You can also cascade them with TD Direct in the upper left and Thermal Desktop in the lower right so that the Meshing Status Dialogue window will be visible to the left. You may also find it convenient to switch between the programs by pressing Alt-Tab. Pressing Alt-Tab will return you to your previous active window, which makes switching between two programs simple and fast. If you are working with a third window, you may need to hold down Alt while pressing Tab a few times to select the right program. Of course, clicking in the correct window always activates it, and you can use the task bar as well.

13. In TD Direct, press **Esc** to ensure nothing is selected. Click on Thermal tab, and select Mesh Ctrl. This will bring up the Default Mesh Controls window. Alternatively, you can right-click in the design window to select Mesh Ctrl.

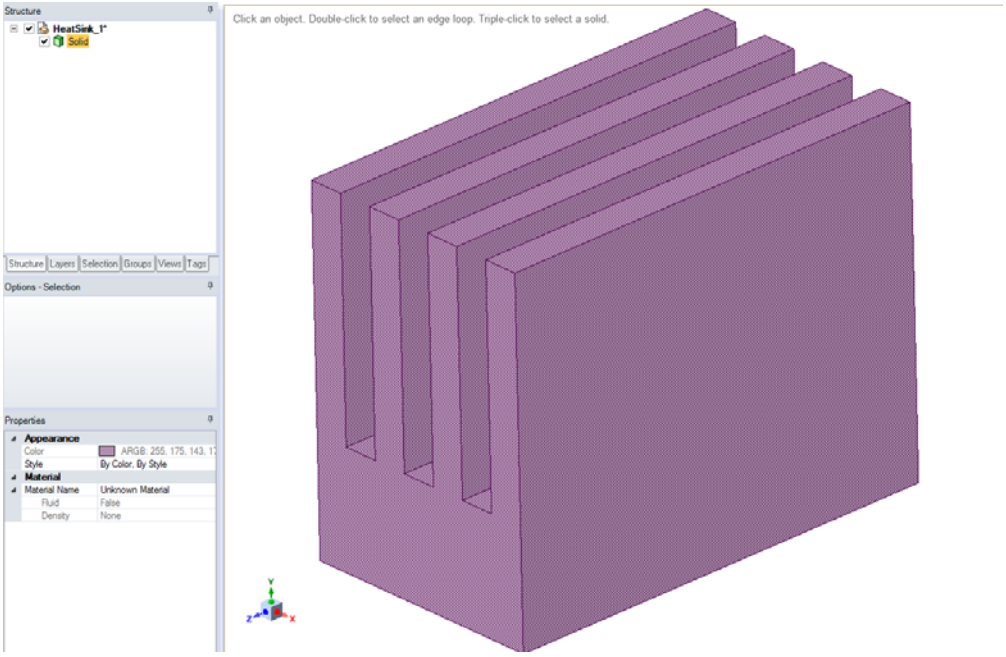


14. The first tab of the Default Mesh Controls is Mesh/Size Type. Change the Relative Mesh Size to **0.07**, which will result in a finer mesh (more nodes). Because no objects are selected, this is the default mesh control that applies to all meshed

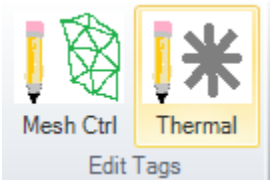
parts. If the background is a pale blue color instead of the peach color, it means you are in local mesh control edit mode (i.e. some part of the object was selected when you clicked Mesh Ctrl). Click Ok.



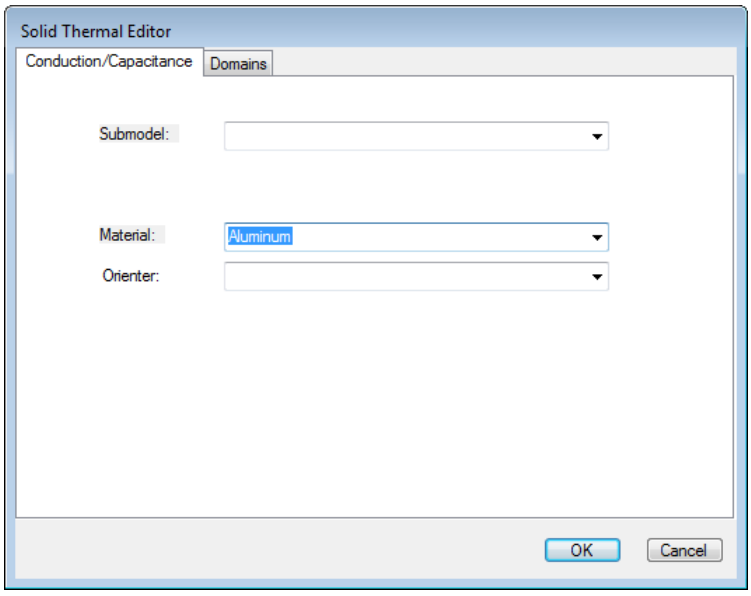
15. Triple-Click the fin to select the solid. Alternatively, you can select the solid in the Structure pane on the left.



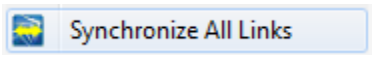
16. Select Thermal in the Edit Tags group.



17. The default submodel is TA_MESH, which can be overridden here. The material can also be selected from the material database that was active in Thermal Desktop during the synchronization between Thermal Desktop and TD Direct. Select the material **Aluminum** from the pull-down menu.



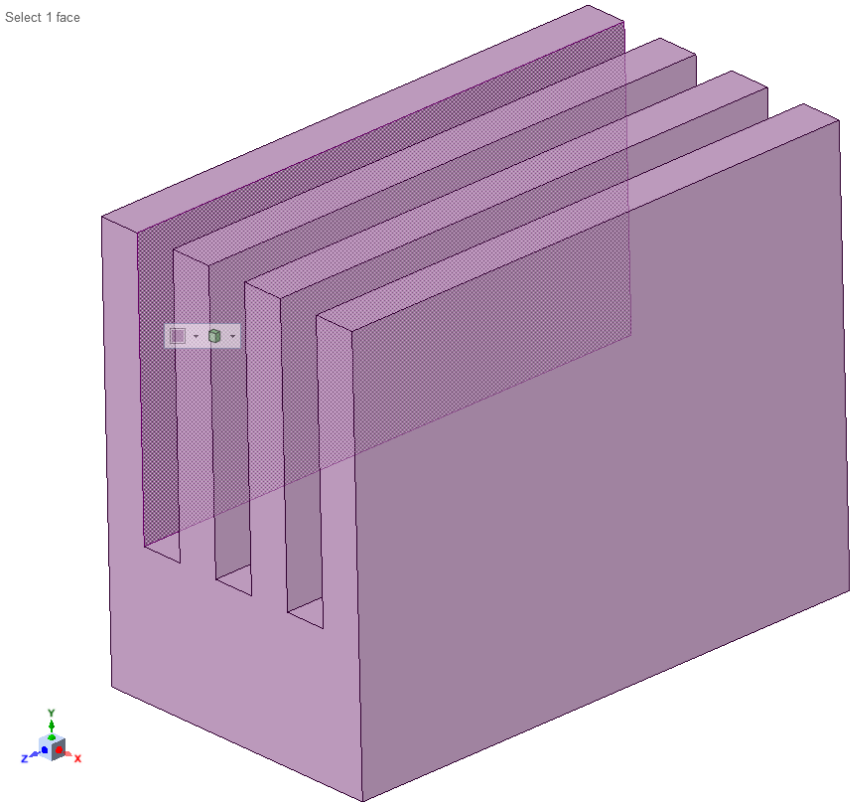
18. Return Thermal Desktop. Use the pull-down menu to select *Thermal > TD Direct > Synchronize All Links*. This will update all TD Direct links in the current drawing. The new mesh should be denser than the first one created.



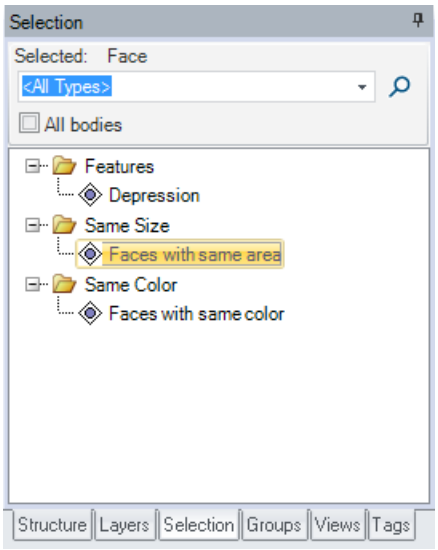
5.1.1 Domain Sets and Domain Tags

The section below will describe how to use Domain tags in TD Direct to generate Domain Sets in Thermal Desktop. Domain tags in TD Direct can be used to mark up parts of the geometry which can then be used in Thermal Desktop as Domain Tag sets. Users may continue from the previous steps or begin with files ***HeatSink_2.dwg*** and ***HeatSink_2.scdoc*** in the *\2_Domains* subdirectory.

19. In TD Direct, press **Esc** to ensure nothing is selected. Then click on one of the large interior surfaces of the depression.



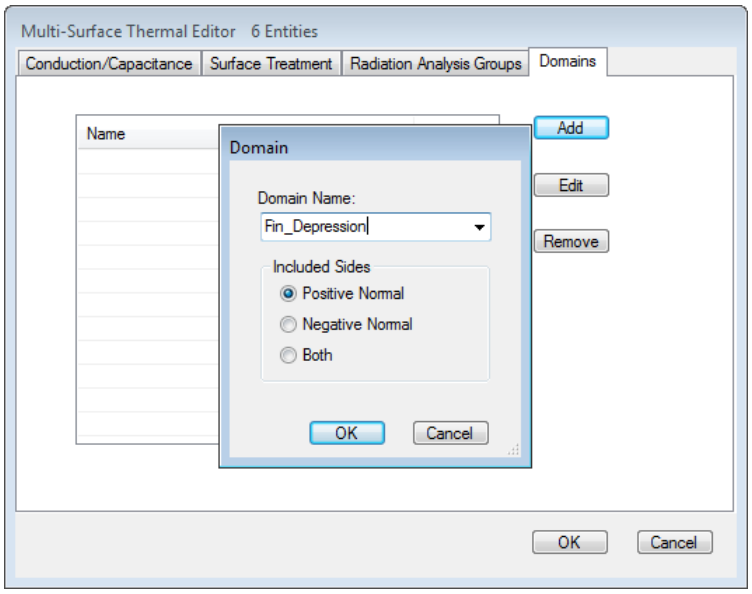
20. Go to the Selection tab in the Structure panel. This is a powerful tool that can help control and assist your selections. In this case, select Faces with same area. That will put all six interior faces into the selection set. This selection tool can be used on many different features such as rounds or holes.



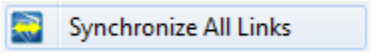
21. Hold down **Ctrl** and select the three narrow faces that make up the bottom of the depression. Using **Ctrl** works the same way it does in most Windows programs by adding to the selection. Now all nine surfaces of the internal depression should be selected.

22. Select Thermal on the Edit Tags group. This is where tags that are not directly related to meshing are applied. An object must be selected to apply a tag in the Thermal editor.

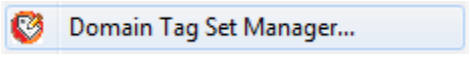
23. Go to the Domains tab, and click on Add. Domain tags can be applied to edges, surfaces, or solids. Add a new Domain tag called “Fin_Depression” to the selected surfaces, and leave the Positive Normal radio button checked. The positive normal direction for a face on a solid is the outward from the visible side. Click Ok to add the domain tag, and click Ok again to exit the Thermal Editor.



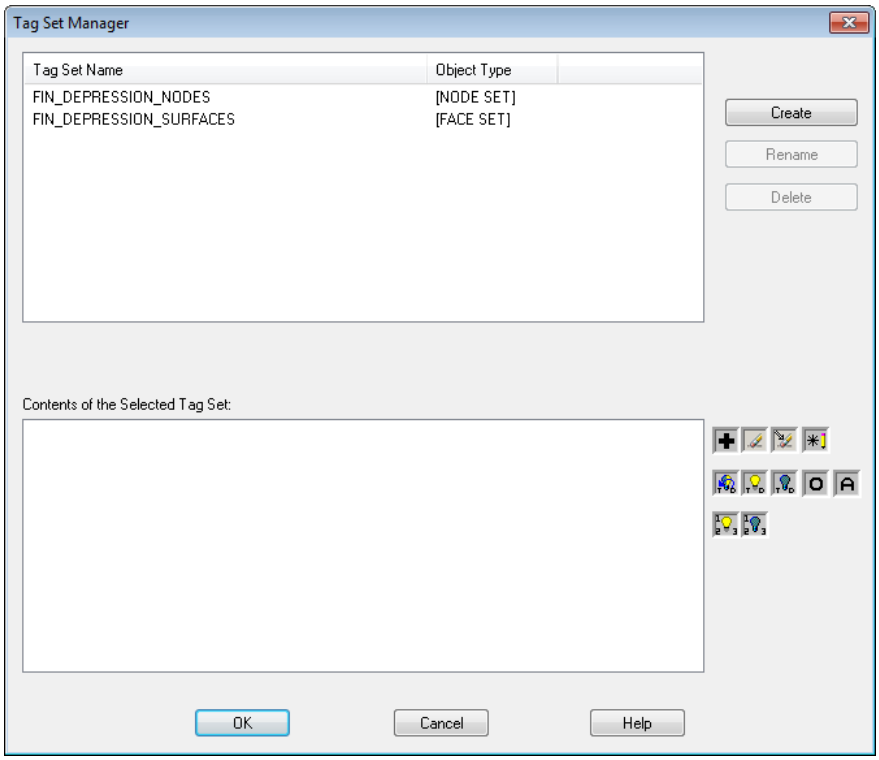
24. Return to Thermal Desktop. Use the pull-down menu to select *Thermal > TD Direct > Synchronize All Links*. This will update all TD Direct links in the current drawing.



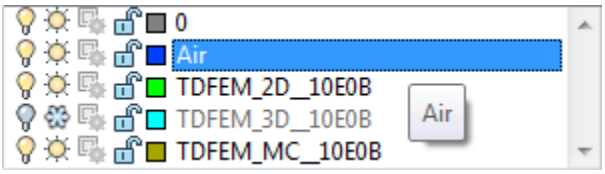
25. Use the pull-down menu to select *Thermal > Domain Tag Set Manager*.



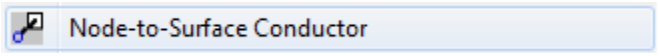
26. The Domain tag “Fin_Depression” that was applied in TD Direct now appears in Thermal Desktop as a Domain tag set. Because the Domain represented surfaces, both surfaces and nodes are available as domain tag sets. Had the domain tag been applied to a solid, there would be domain tag sets for the solid, the surfaces, and the nodes. There is no need to change anything here, so simply click Ok to close the window.



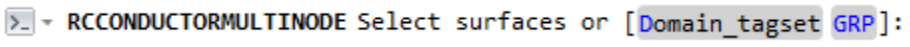
27. Click on the down arrow next to the Layer control and select Air to set it as the current layer.



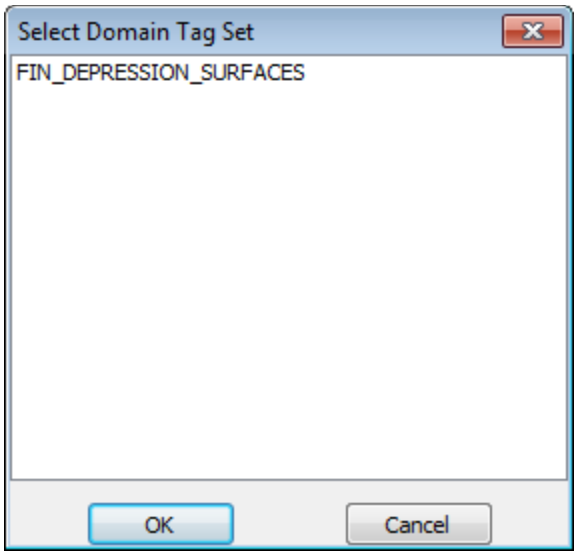
28. Use the pull-down menu to select *Thermal > FD/FEM Network > Node-to-Surface Conductors*.



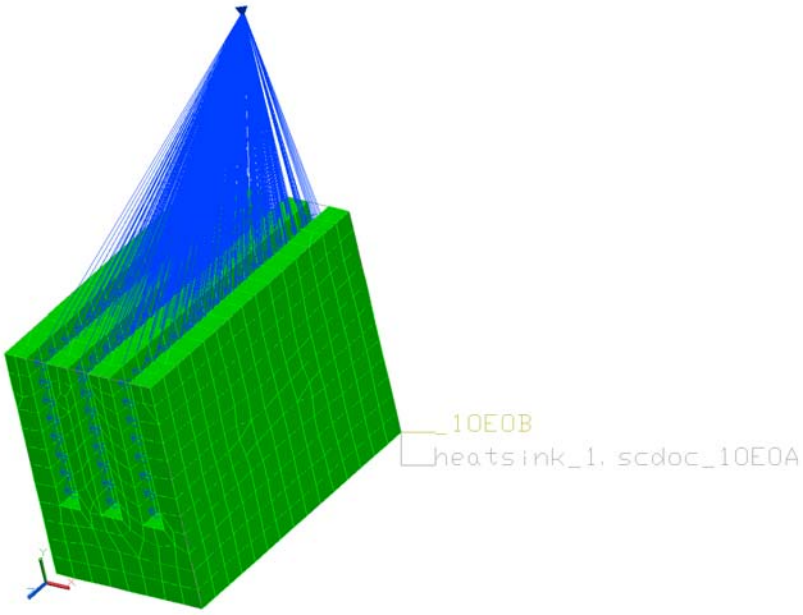
29. The command line will prompt you for a node. Click on the single Air node. The second selection is not an actual surface, but the Domain tag set. Either type “D” in the Command Line and press enter, or simply click on “Domain_tagset” in the Command Line Prompt.



30. The list of available and appropriate Domain tag sets will open, which right now is only the Fin_Depression_Surfaces. Select it and click OK.

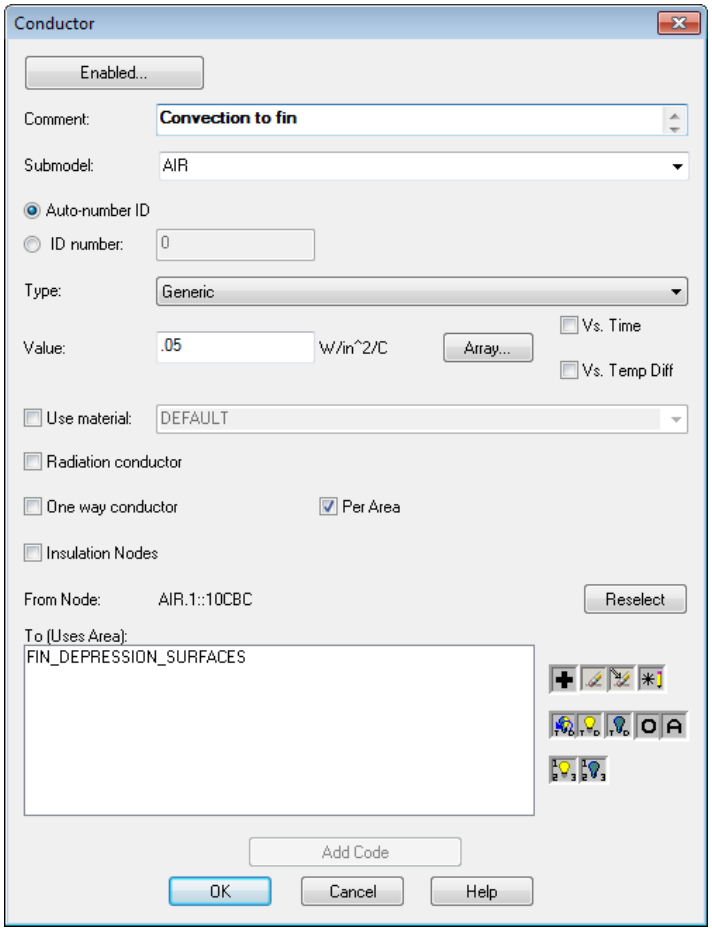


31. Press **Enter** to complete selecting surfaces. You have now created a conductor between a node and a domain tag set. The effect is to connect the node to all the surfaces that are associated with that Domain tag set. If all the surfaces in the Domain tag set are deleted, the conductor will remain (though not generate any conductors in SINDA). If new surfaces are added to the domain tag set, they will also be part of the conductor.

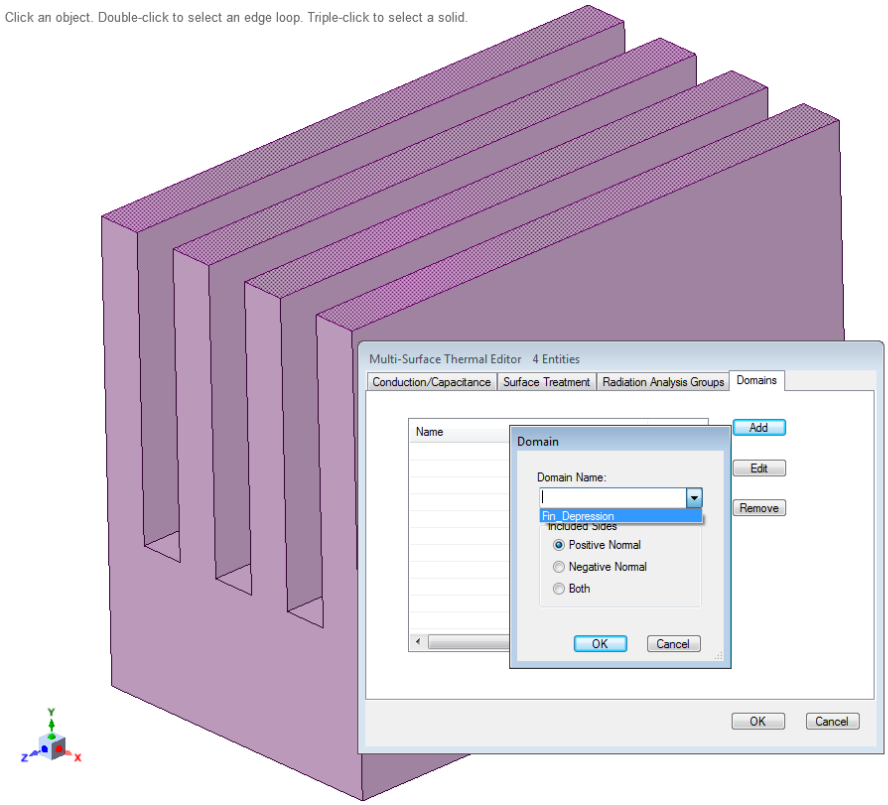


32. Click on the newly created conductor and select *Thermal > Edit*.

33. Change the Submodel to **Air**. Check the box for Per Area and change the Value to **0.05** W/in^2-C. Add a comment indicating this is convection to the fin. Click Ok to finish editing the conductor.

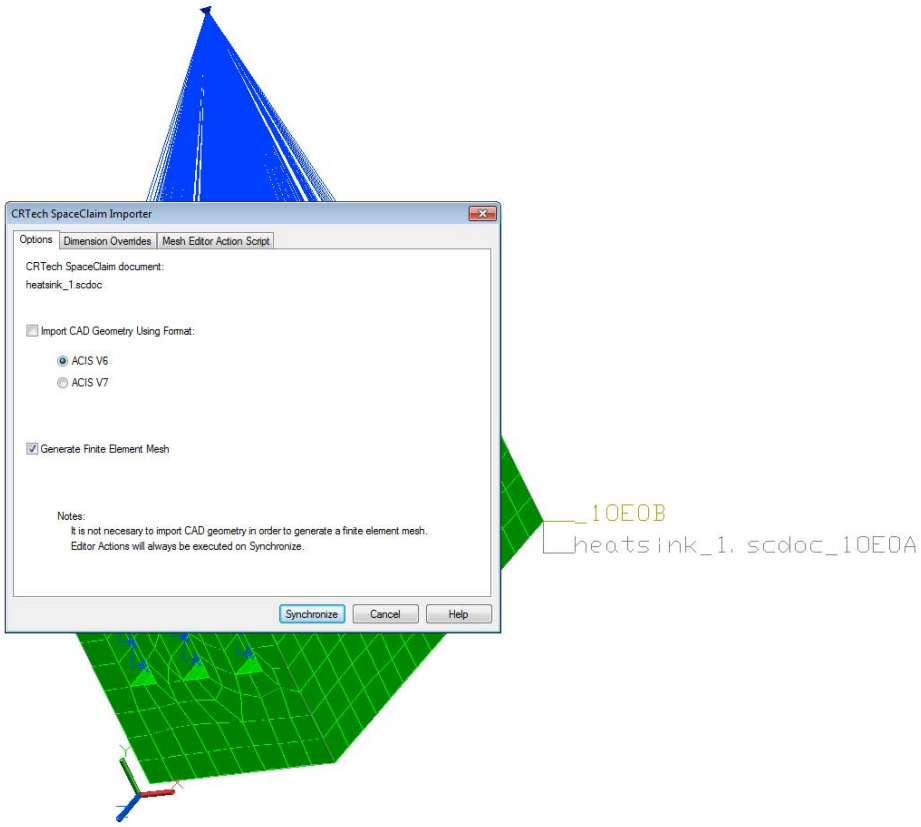


34. Return to TD Direct. Hold down **Ctrl** to select all four tops of the fins. These surfaces will be added to the domain. Go to the Thermal Ribbon and select Thermal in the Edit Tags group. Go to the Domains tab and select Add. From the drop down menu, select Fin_Depression. Click Ok, and Click Ok again to close the Thermal Editor.

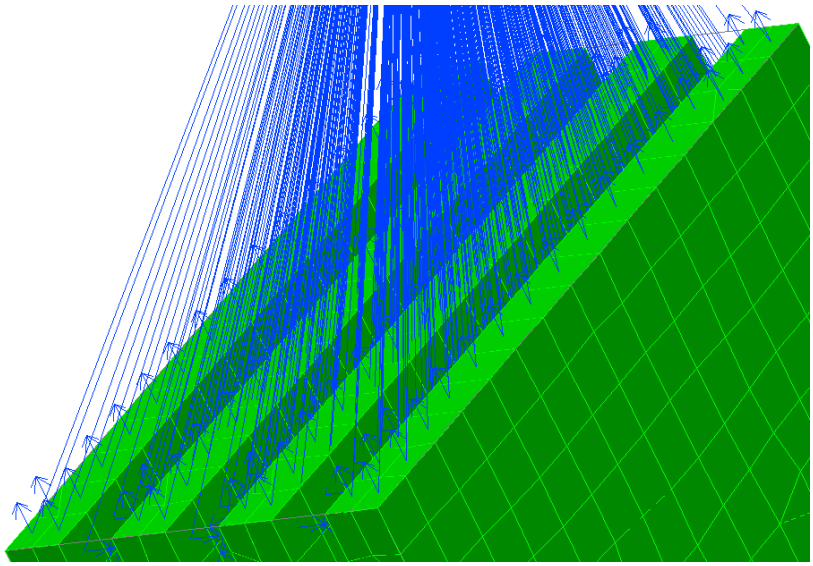


35. Return to Thermal Desktop, and select the Mesh Importer by clicking on the text in the graphics area that shows the file name of the imported TD Direct document. Edit the importer by using the pull down menu to select *Thermal > Edit*.

36. If there are several importers active, this menu allows synchronizing only one of them at a time. Click Synchronize to update the new surfaces into the domain tag set Fin_Depression.



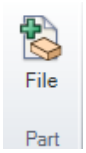
37. The newly updated mesh will show the convection is now also attached to the top of the fins.



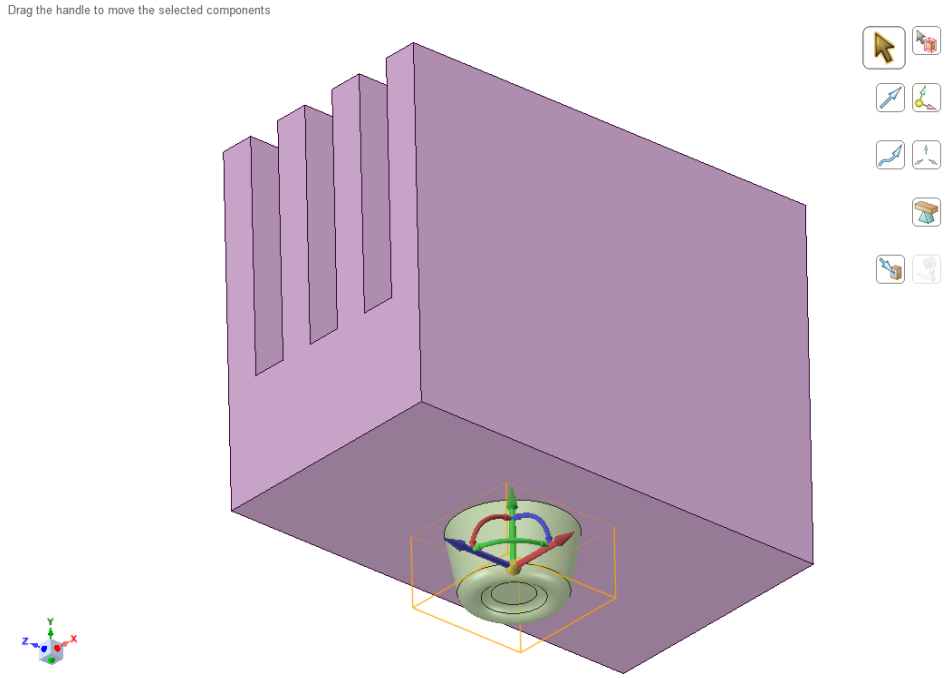
5.1.2 Mesh Interfaces

When two separate components in a mesh are touching, there are three separate ways to control the interface. The components can be merged into a single body, they can be meshed to make for precise contact in Thermal Desktop, or they can be meshed completely separately. This section will cover how to configure them for precise contact in Thermal Desktop, but the steps are very similar for the other two. Users can continue from previous steps or start with files ***Diode.scdoc***, ***HeatSink_3.dwg*** and ***HeatSink_3.scdoc*** in the *\3_Mesh_Interfaces* subdirectory.

38. In TD Direct, go to the Insert tab and select File.

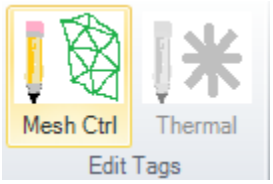


39. Select the document called ***Diode.scdoc***. The new component will be inserted on the bottom of the heat sink. Press **Esc** to exit the Move command that is automatically brought up. When you do your own work, you will likely have to move the parts when you import them.

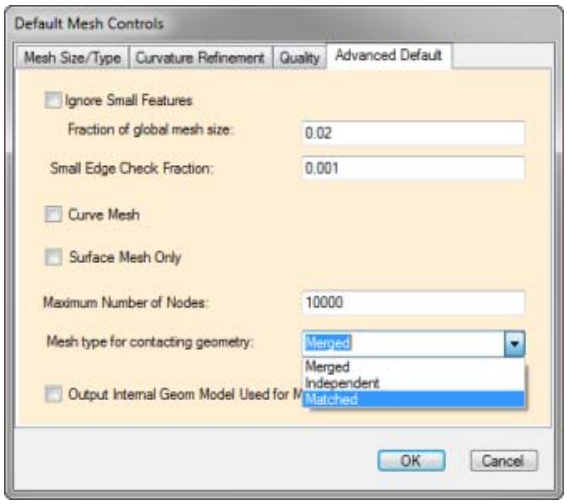


40. In the structure tree these are visibly two separate components. The diode and the heat sink will create separate meshes, and the user can control how the meshes interact. Press **Esc** to ensure nothing is selected, and then select Mesh Ctrl on the

Thermal tab.

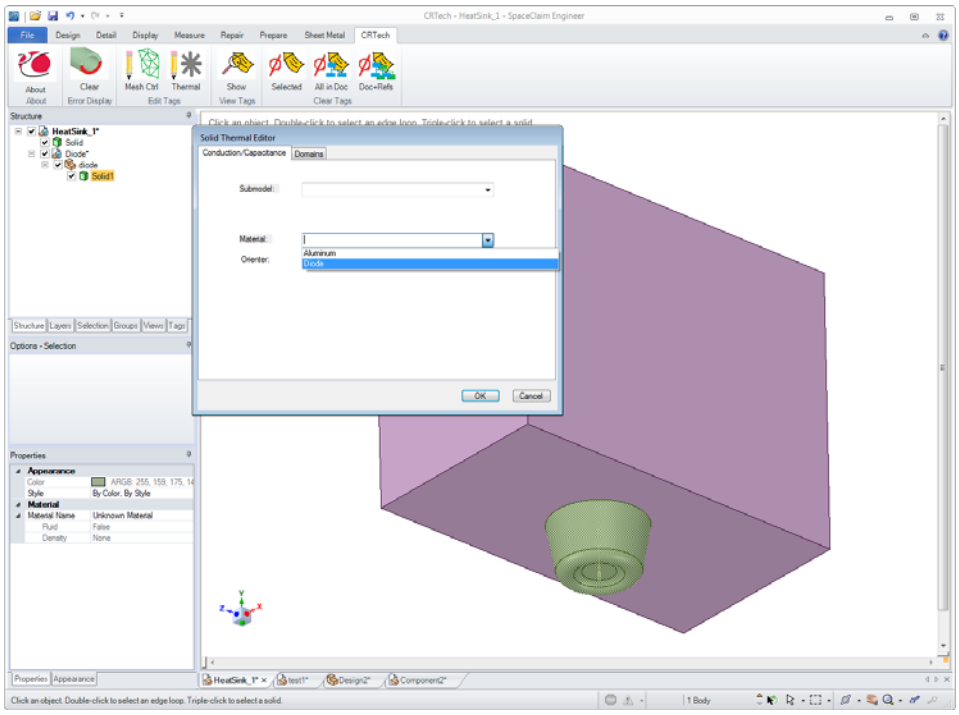


41. Select the Advanced Default tab and change Mesh type for contacting geometry to Matched. The Independent option will create two meshes that do not have any relation to each other. The Merged option will make one continuous mesh as if it were a single material. The Matched option makes two meshes on either side of the interface matched identically. All elements and nodes are overlaid precisely. Click Ok to close the window.

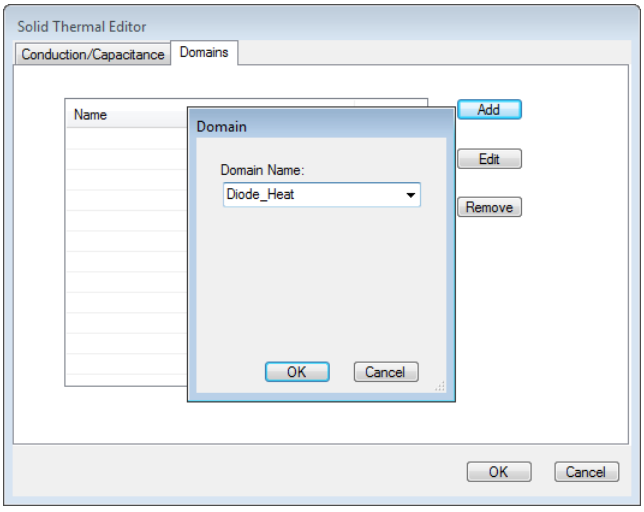


42. Triple click on the diode. Triple clicking selects the solid. Alternatively, you can click on the solid under Diode in the structure tree. On the Thermal tab, in the Edit Tags group, select Thermal.

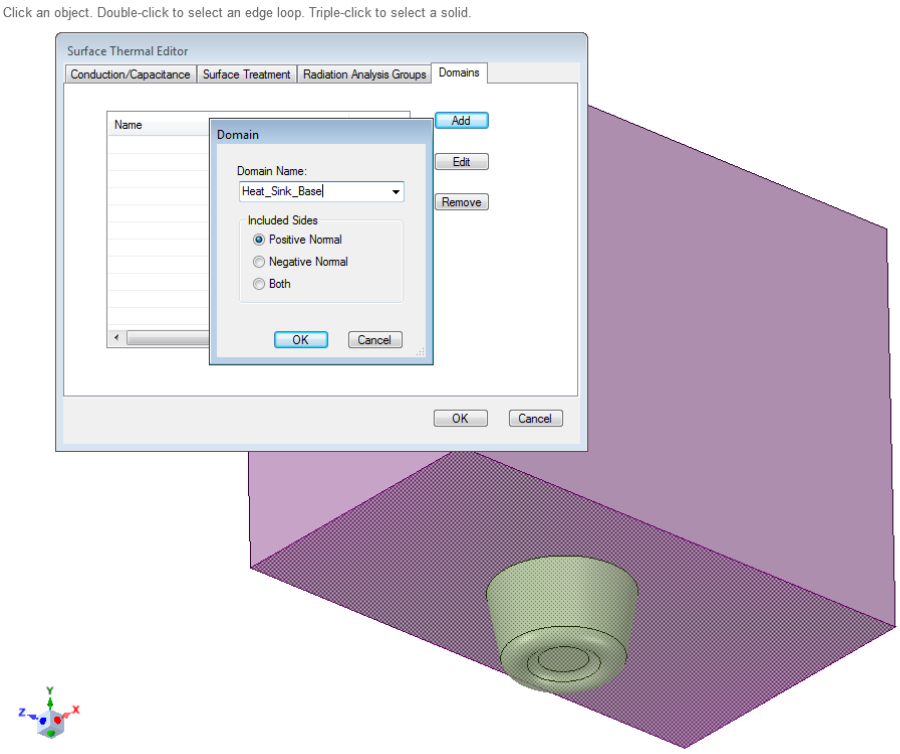
43. On the Conduction/Capacitance tab, select “Diode” for the material of the diode. The properties for this material are dummy properties useful only for this tutorial.



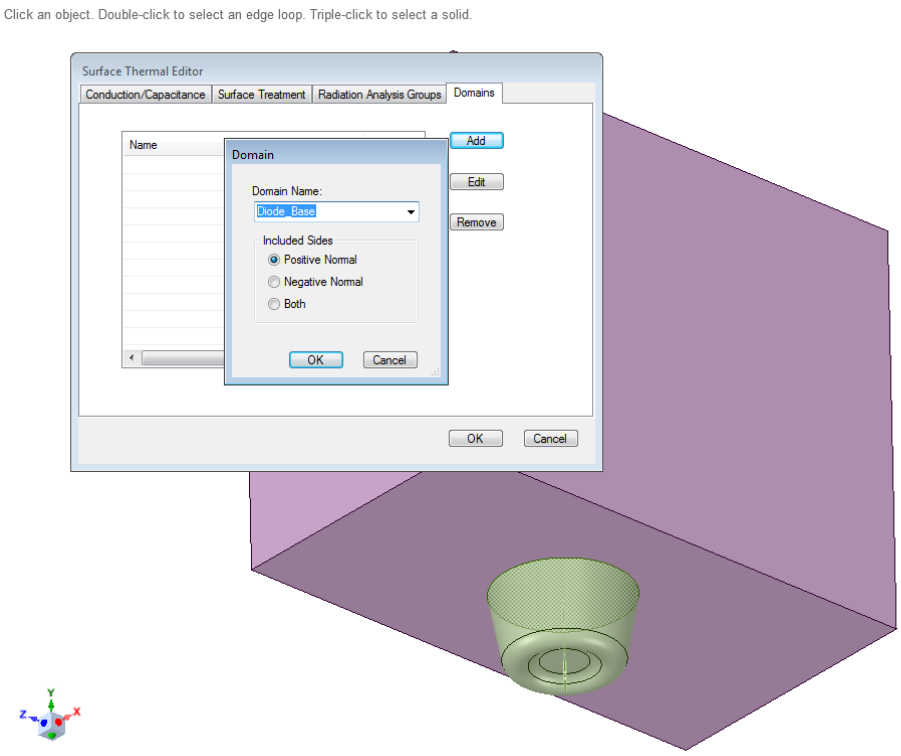
44. On the Domains tab, Add a new domain tag to the solid called “Diode_Heat.” This will be used to apply a heat load to the solid in Thermal Desktop. Click Ok to close the Solid Thermal Editor.



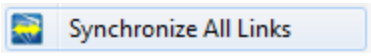
45. In order to get matching surfaces elements in Thermal Desktop for the contactor between diode and the heat sink, the “from” and “to” surfaces in TD Direct must each be assigned domain tags. Press **Esc** to ensure nothing is selected, and then click on the bottom face of the heat sink. Select the Thermal tag editor. Go to the Domains Tab and Add a Domain tag named “Heat_Sink_Base.” Click Ok to close the Surface Thermal Editor.



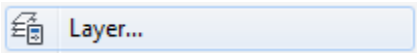
46. Move the cursor over the diode and scroll the mouse wheel until the flat circle of the diode that contacts the fin base is highlighted. Select that surface with a click. Use the Thermal tag editor and add a Domain tag called “Diode_Base.” Click Ok to close the Surface Thermal Editor.



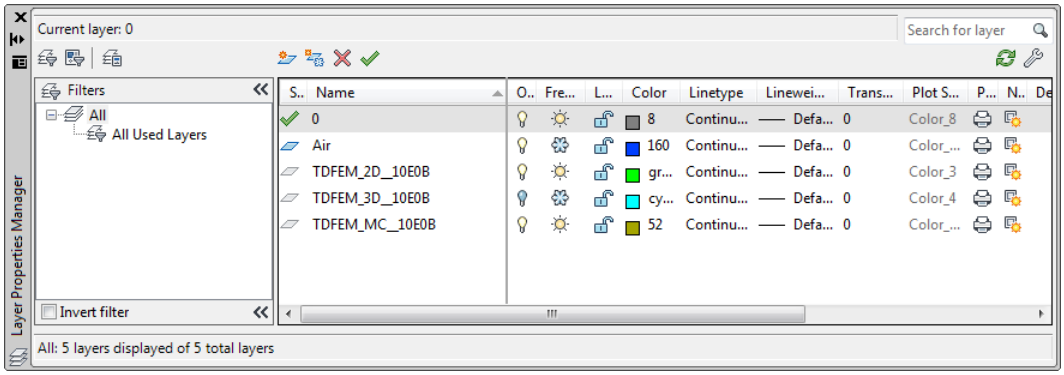
47. Return to Thermal Desktop and use the pull-down menu to select *Thermal > TD Direct > Synchronize All Links*.



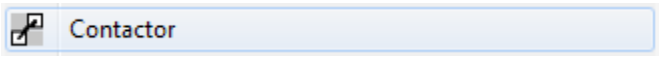
48. Use the pull-down menu to select *Format > Layer* to enter the Layer Manager.



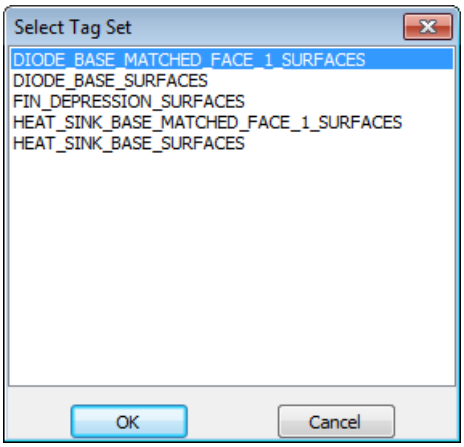
49. Set the current layer to 0 by highlighting it and clicking on the green checkmark. Freeze the Air layer by clicking on the icon of the sun so it turns into a snowflake. Click the X in the upper left or right corner to close the Layer Manager.



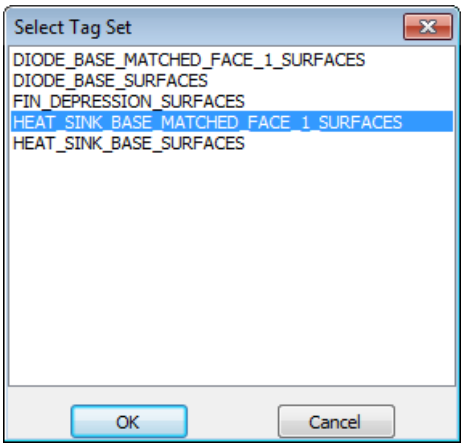
50. Use the pull-down menu to select *Thermal > FD/FEM Network > Contactor*.



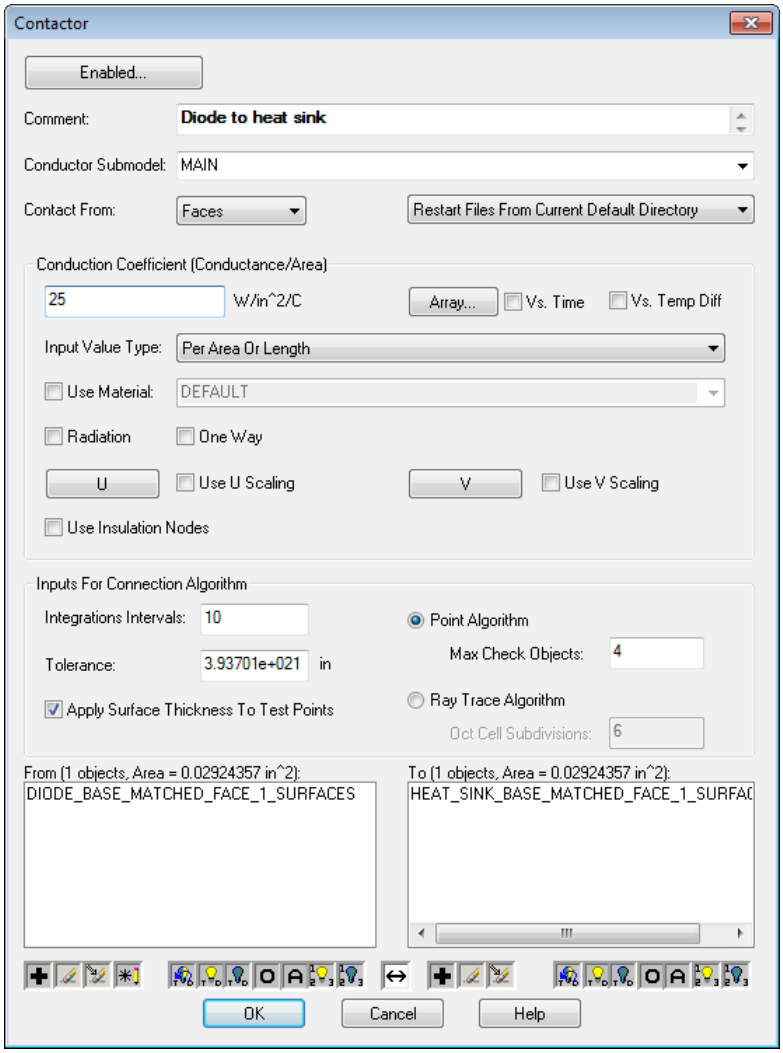
51. The command line will prompt you for “From objects.” Click on the box “Domain_tagset” in the command line or simply type D followed by enter. There are many more Domain tag sets available now. This contactor is between the diode and the heat sink, so select DIODE_BASE_MATCHED_FACE_1_SURFACES. In this particular case, this is the same set of surfaces that make up all of DIODE_BASE_SURFACES because the entire surface is in contact with the heat sink. Click Ok, and press Enter to complete selecting From surfaces.



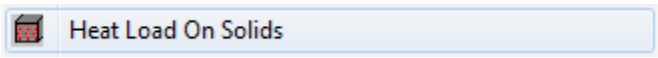
52. The command line will prompt you for “To objects.” Again, click on the box “Domain_tagset” in the command line or simply type D followed by enter. Select HEAT_SINK_BASE_MATCHED_FACE_1_SURFACES. This is the subset of HEAT_SINK_BASE_SURFACES that is in contact with the diode. By using matching domain tag sets, a precise area for conduction is ensured. Click Ok, and press Enter to complete selecting To Objects.



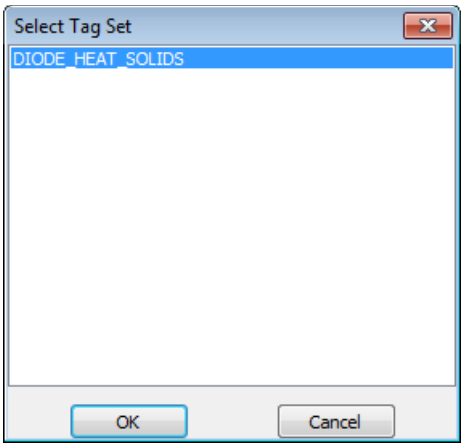
53. The contactor is the same as any other you may have created except it is between two domain tag sets rather than the surfaces directly. The domain tag sets in Thermal Desktop are automatically filled with the element surfaces that are generated from the CAD surfaces in TD Direct. Add a comment indicating this is between the diode and the heat sink and change the Conduction Coefficient to **25** W/in^2-C. Click Ok to close the Contactor window.



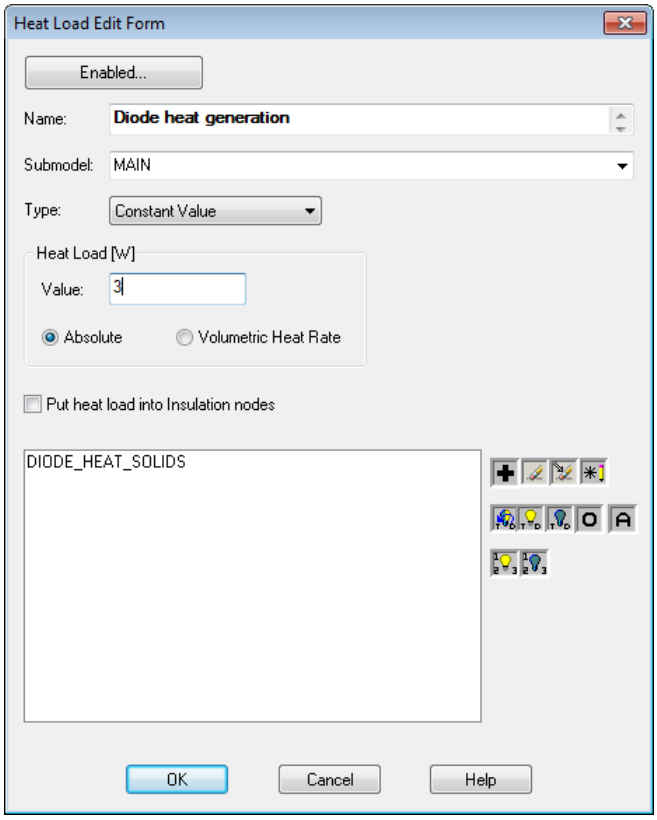
54. Use the pull-down menu to select *Thermal > FD/FEM Network > Heat Load on Solids*.



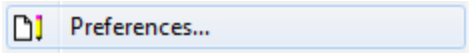
55. The command line will prompt for a solid or domain tag set. Choose "Domain_tagset," and select DIODE_HEAT_SOLIDS. This represents the entire solid of the diode. Click Ok and press Enter to complete selecting solids.



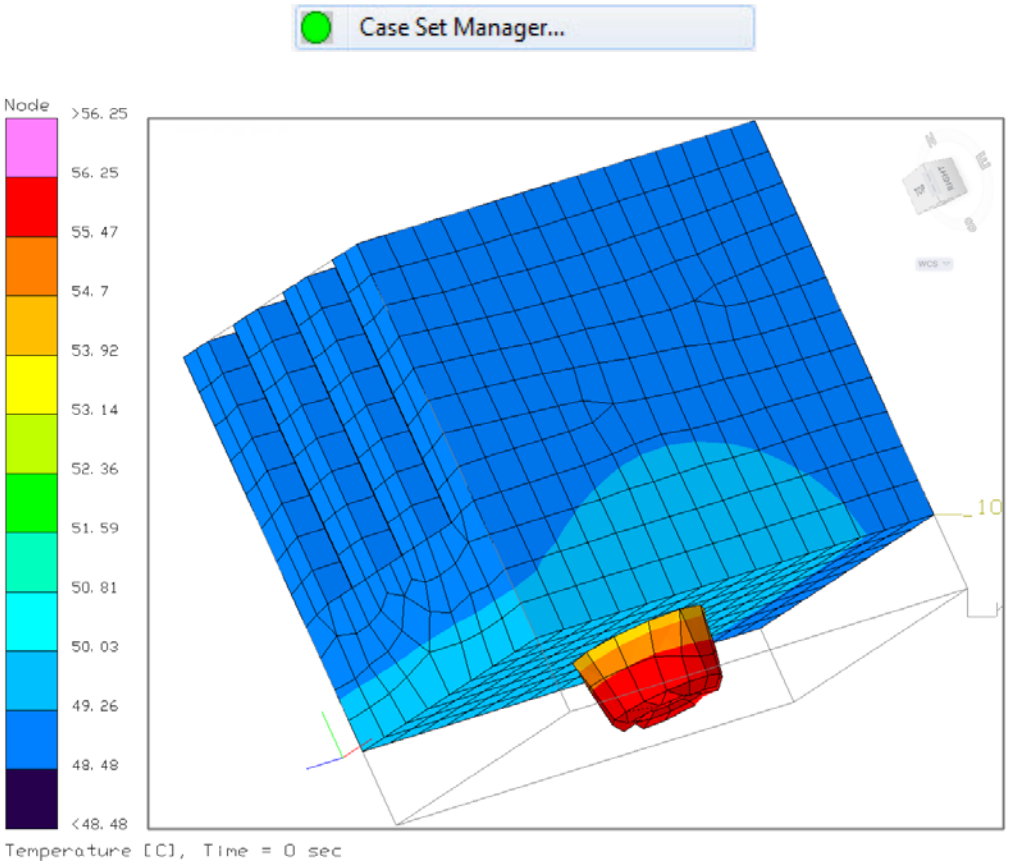
56. Add a comment indicating this is the heat applied to the diode. Change the value of the Heat Load to **3**W.



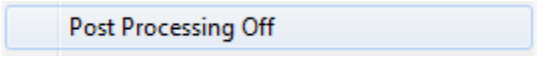
57. Use the pull-down menu to select *Thermal > Preferences*. On the Graphics Visibility tab, deselect Heat Loads / Heaters / Pressures. This will turn off the visibility of the heat source markers on the diode.



58. If you have not done so already, be sure to save the file. Then use the pull-down menu to select *Thermal > Case Set Manager*. Run the default case, which is steady state.



59. Use the pull-down menu to select *Thermal > Post Processing > Post Processing Off*.



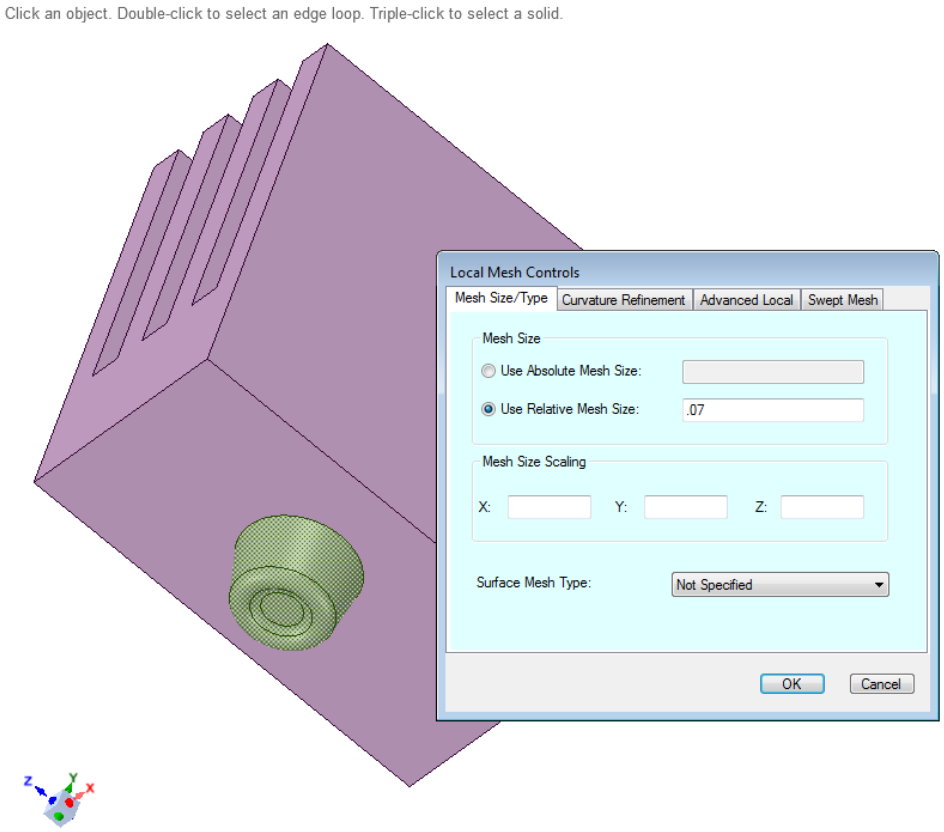
5.1.3 Local Mesh Controls

The default mesh controls are applied to all objects unless they are overridden by local mesh controls. Local mesh controls can only be used to refine the nodal resolution. A local region cannot be made coarser than the global specification. Edges, surfaces, and solids can be given specific mesh controls to precisely control the desired nodal resolution, use a coarse global specification and refine as necessary. For example, it may be necessary to have higher resolution in an area of high thermal gradients or to accurately model a curved surface. Users can continue from previous steps or start with files ***Diode.scdoc***, ***HeatSink_4.dwg*** and ***HeatSink_4.scdoc*** in the *\4_Local_Mesh* subdirectory.

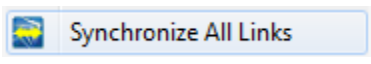
60. In TD Direct, select the solid of the Diode either by triple clicking or selecting it from the structure tree. The following steps will increase resolution of the diode mesh.

61. Select Mesh Ctrl (right-click or in the Edit Tags group). Note that the background of the window is blue and the window title is “Local Mesh Controls.”

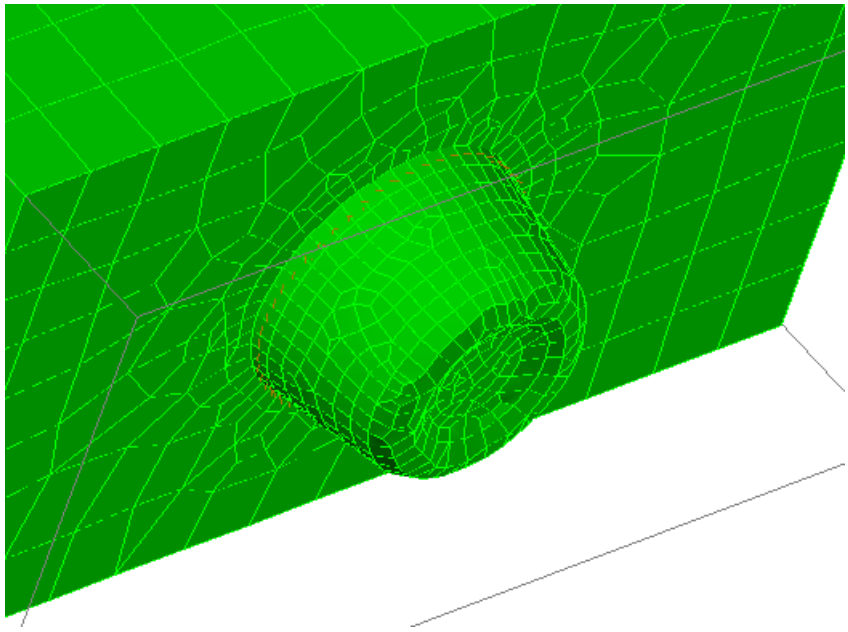
62. On the Mesh Size/Type tab, the Use Relative Mesh Size is blank. Any blank entries will use the default mesh controls. The mesh size is relative to the bounding box around the object selected. Type **0.07** in the Relative Mesh Size box. This is the same value used in the default mesh controls, however, a different mesh size will be generated.



63. Go to Thermal Desktop and use the pull-down menu to select *Thermal > TD Direct > Synchronize All Links*.



64. Note the extremely fine mesh generated on the diode even though the local and default mesh controls were set to the same relative value. For the default mesh controls, the initial mesh size was 0.07 of the largest dimension of the bounding box of the entire assembly. For the diode's local mesh control, the initial mesh size was 0.07 of the largest dimension of the bounding box of the diode.

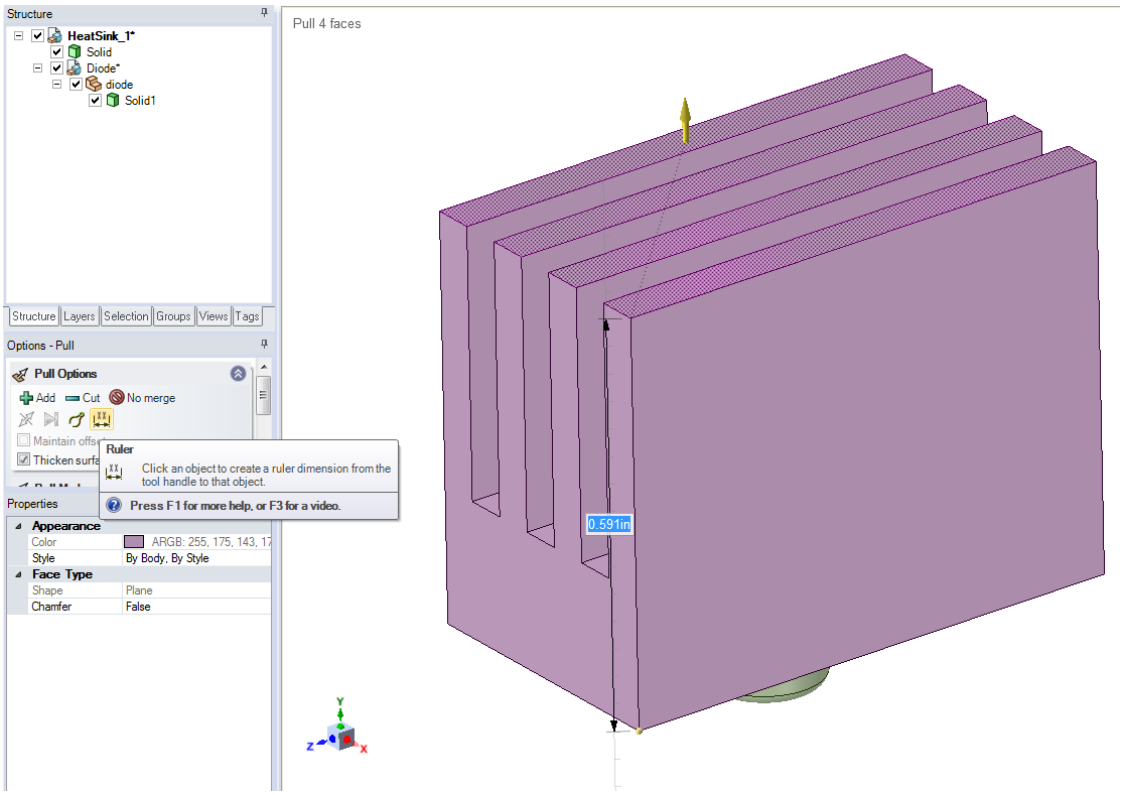


65. Return to TD Direct edit the local mesh control of the diode solid again. Change the relative mesh size to 0.10 and synchronize. The new mesh should be more appropriate.

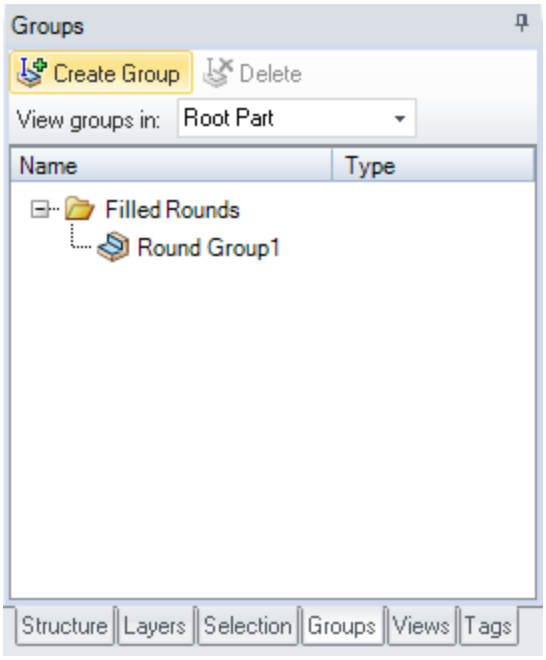
5.1.4 Driving Dimensions

Driving Dimensions are created in TD Direct and allow the user and Thermal Desktop to change certain dimensions of the objects in TD Direct, which in turn cause the mesh to be updated. This capability can allow for a rapid parametric study of the design.

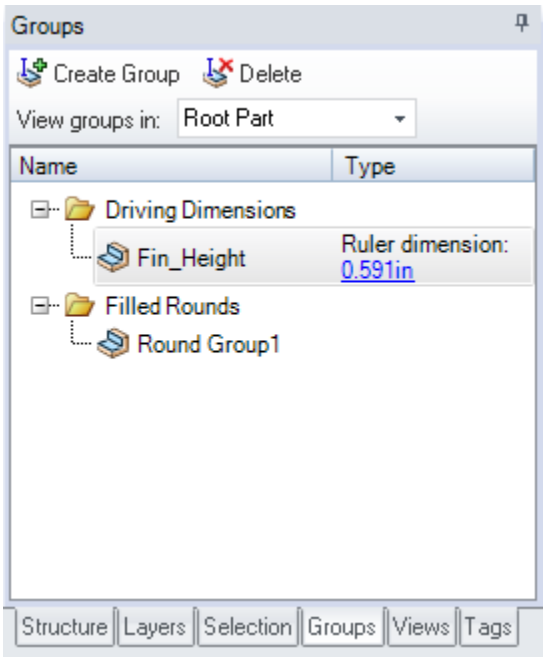
66. In TD Direct, select Pull on the Design tab. While holding down **Ctrl**, select the four tops of the fins.
67. Under Pull Options on the left panel, select the Ruler. On the object, two points will appear, one marking the surfaces you are pulling and the other marking some other point. Drag the bottom marker to a bottom edge or corner of the fin. This will define the total fin height.



68. With the value of 0.591in still highlighted, select the Groups tab on the Structure panel. You will see a Group already exists, Filled Rounds. Select Create Group.

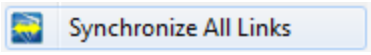


69. A group of Driving Dimensions will be created called Group1. Click on Group1 twice to rename it “Fin_Height”.

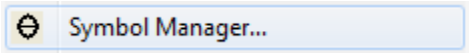


70. Press **Esc**. Click on the value 0.591in and enter a new value of 0.4. The fin height of the TD Direct model will change accordingly.

71. In Thermal Desktop, use the pull-down menu to select *Thermal > TD Direct > Synchronize All Links*. The mesh will be regenerated with the smaller fins.

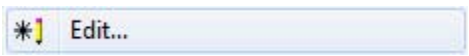


72. Select *Thermal > Symbol Manager*.



73. Create a new symbol by typing “FinHeight” in the box for New Symbol Name and then clicking Add. Give this symbol a value of 0.7. Click Ok to close the Expression Editor, and click Done to close the Symbol Manager.

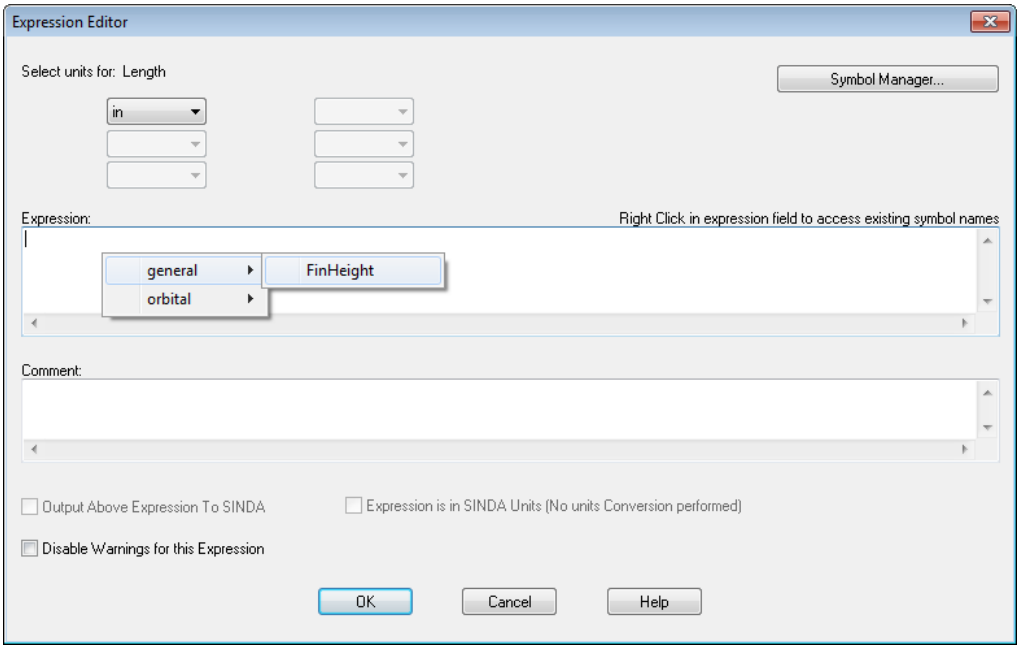
74. Select the TD Direct Importer anywhere on the bounding box or the text that shows the file name of the TD Direct document. Then use the pull-down menu to select *Thermal > Edit*.



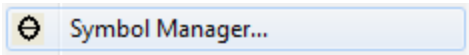
75. Go to the Dimension Overrides tab. From SC Group Name, select Fin_Height and then click Add. The driving dimension created in TD Direct can also be controlled from here. You could input a single value, but it is typically more useful to input a

symbol. Double click in the text box that says “0.4” to bring up the Expression Editor.

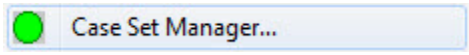
76. In the Expression Editor window, right click in the Expression box. Select General and FinHeight. The height of the fin in TD Direct is now controlled by the symbol FinHeight. Every time the symbol FinHeight is changed, the mesh will automatically update. Click Ok to close the Expression Editor, click Ok to close the Edit Driving Dimension window, and click Synchronize.



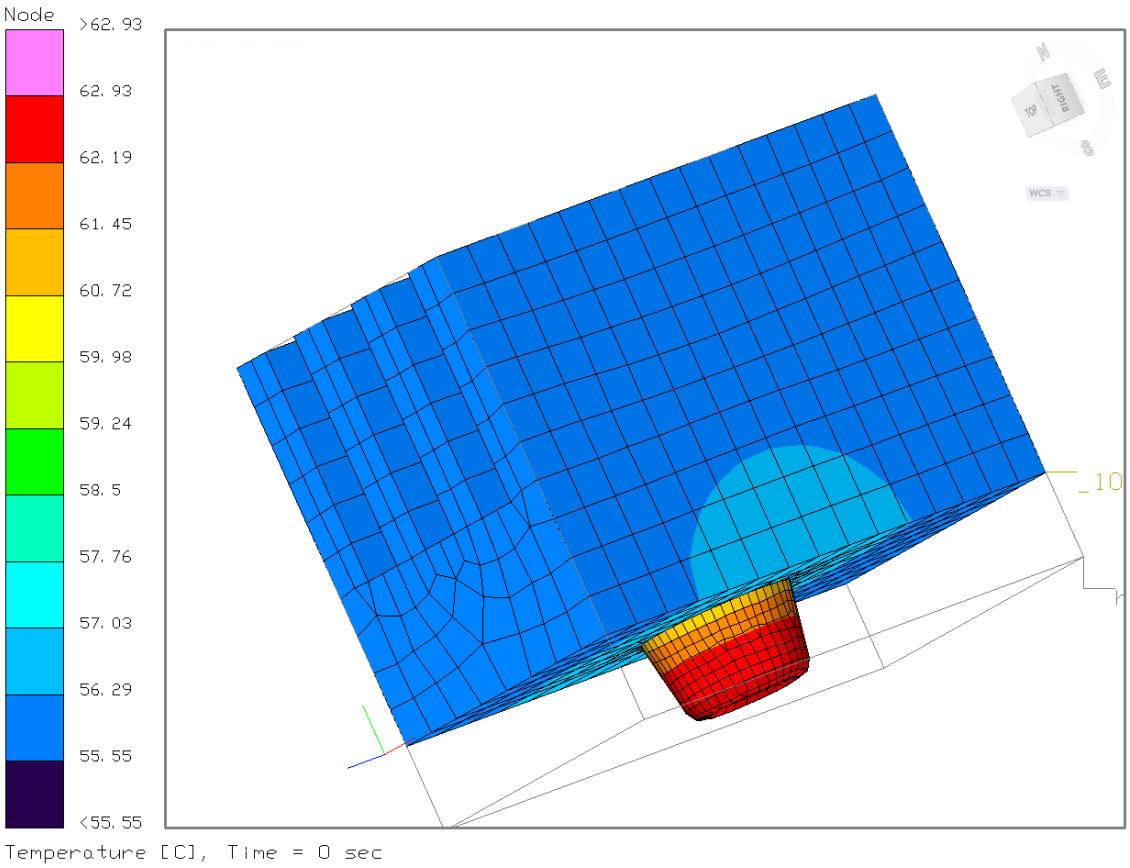
77. Go into the Symbol Manager and change the value of FinHeight to 1.0. When you close the Symbol Manager, the mesh will automatically update.



78. Use the pull-down menu to select *Thermal > Case Set Manager*. Run the default case.



79. Look at the temperature of the diode, and then change the symbol FinHeight to 0.5. Return the case and compare the results.


Node
>62. 93
62. 93
62. 19
61. 45
60. 72
59. 98
59. 24
58. 5
57. 76
57. 03
56. 29
55. 55
<55. 55
10
Temperature [C], Time = 0 sec

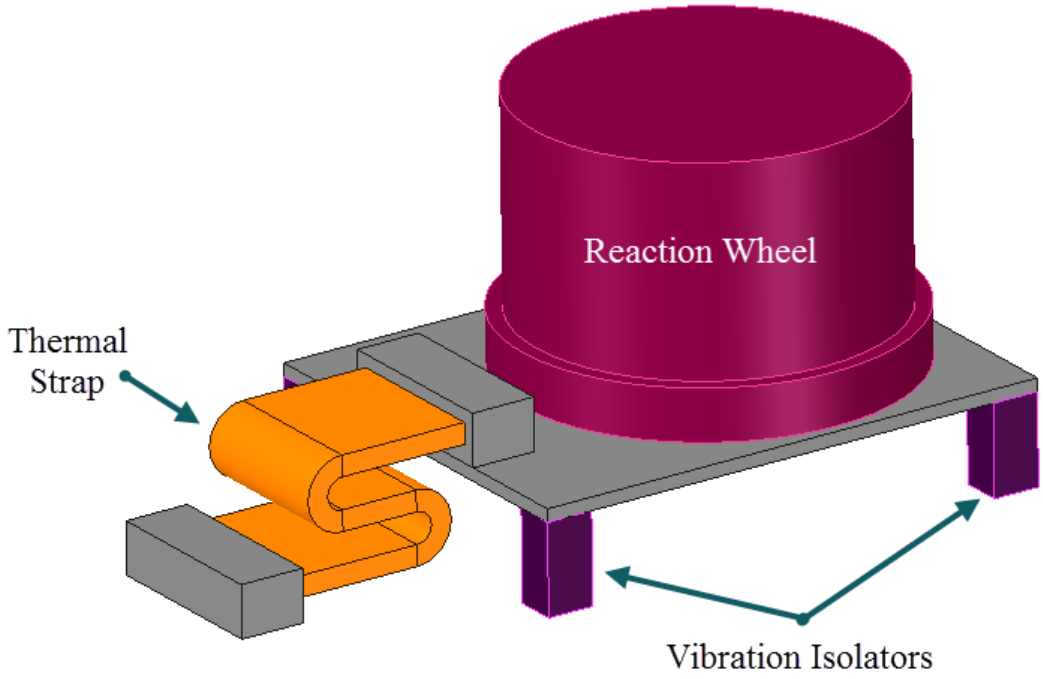
5.2 Swept Mesh Tutorial

This tutorial focuses on the swept mesh methods available in TD Direct. The geometry has already been created, and the tutorial will step through the tags required to generate the correct mesh, assign properties, and run the analysis.

This tutorial will follow a hypothetical problem scenario that may be encountered by a thermal engineer. A reaction wheel, which is a device that may be used to rotate a spacecraft about an axis by speeding up or slowing down, is initially mounted to a spacecraft panel. This can introduce vibrations into the main body of the spacecraft, which can cause a problem for optics on the spacecraft. In this scenario, it has already been determined that the reaction wheel must be mounted to a separate plate which will then be mounted to the panel using tuned vibration isolators. Vibration isolators typically do not transfer heat well, and this may cause the reaction wheel to overheat. A parallel path for heat needs to be used. This will assuredly transfer some vibration, but ideally a middle ground can be found where enough heat is transferred away from the reaction wheel and the vibration requirement is satisfied.

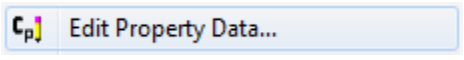
The parallel path for heat is a thermal strap. It is a stack of very thin copper sheets merged into aluminum blocks at the ends. Because the copper is so thin, it is flexible and does not transfer much vibration. While another team determines whether this solution is viable from a vibrations standpoint, the thermal engineer must determine whether it meets the thermal requirements.

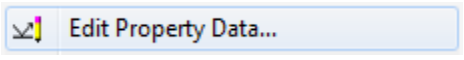
For the purpose of this tutorial, several simple assumptions will be used that are neither realistic nor conservative. First among them is the assumption of perfect contact between all components.



1. Open ***ReactionWheel_1.scdoc*** in the *\Tutorials\TD Direct\Swept* subdirectory. Look at the Structure tree on the Structure Panel and note the assembly has been broken into several components. The thermal strap itself is broken into 5 components, which is done to prevent the solids from combining into a single solid in case the user accidentally pulls them incorrectly. The strap must be in five solids to simplify the shapes.

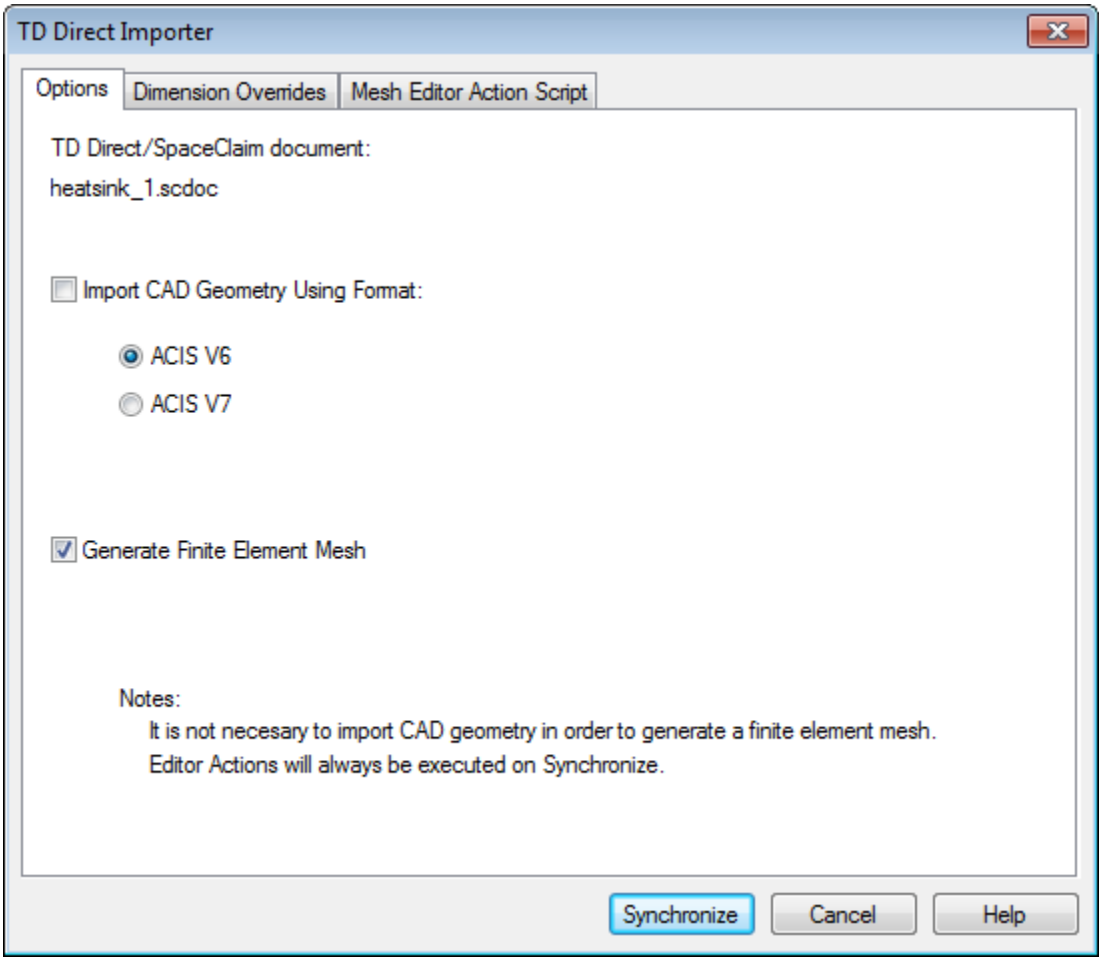
2. Open ***ReactionWheel_1.dwg*** in Thermal Desktop. This file is essentially empty, but it is tied to the optical and thermophysical property databases in the same directory. Edit the thermophysical property (*Thermal > Thermophysical Properties > Edit Data*) and optical property (*Thermal > Optical Properties > Edit Data*) databases to ensure they have been loaded. The thermophysical database should show properties for Aluminum, Copper, and Isolator. The Optical database should show properties for Black Anodize, Copper, and Isolator. If these properties are not found, the database files are not in the current working directory.



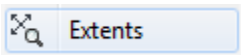


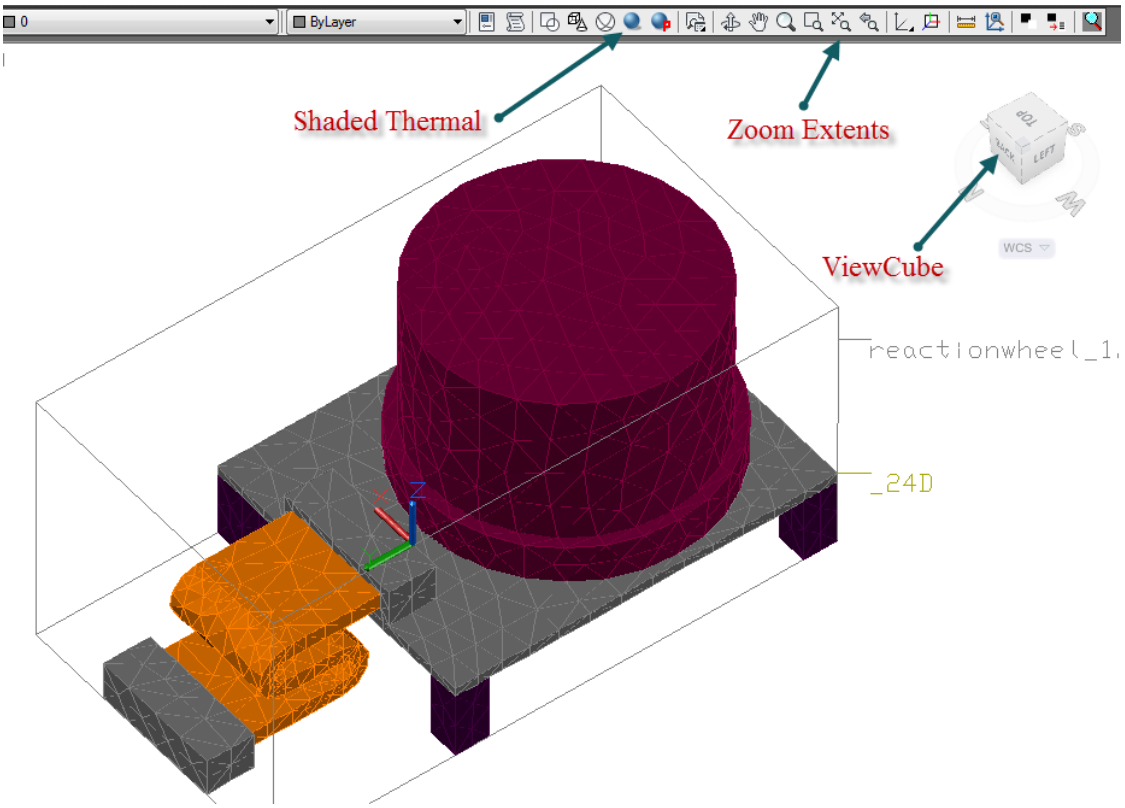
3. In Thermal Desktop use TD Direct to create a link to the TD Direct document. Select *Thermal > TD Direct > Create Link*. Choose ***ReactionWheel_1.scdoc***.

4. Deselect Import CAD Geometry and Select Generate Finite Element Mesh. Click Synchronize to create the default mesh in Thermal Desktop. It is usually best to start with a simple mesh and then incrementally make changes. If there are problems generating the mesh, it will be easier to determine what is causing those problems if it is tested incrementally. Another benefit of establishing the link is it will import the names of the thermophysical and optical properties into TD Direct.

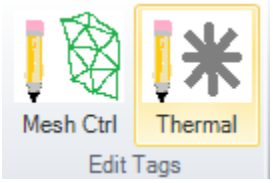


5. You may have to select Zoom Extents (*View > Zoom > Extents* or using the toolbar shortcut) several times to find the imported mesh. Select the view style Shaded Thermal to see the mesh as a solid. Rotate the view using the ViewCube to verify the mesh imported correctly. This is simply a starting point to begin developing the desired mesh.

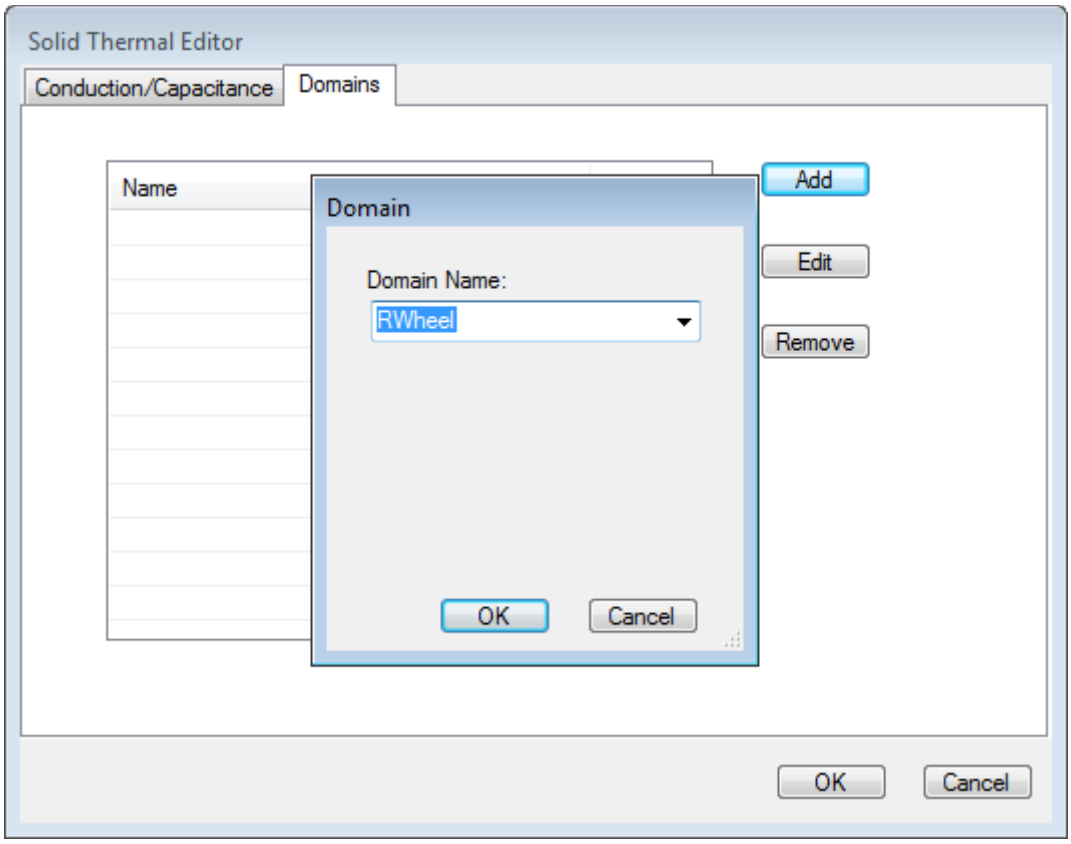




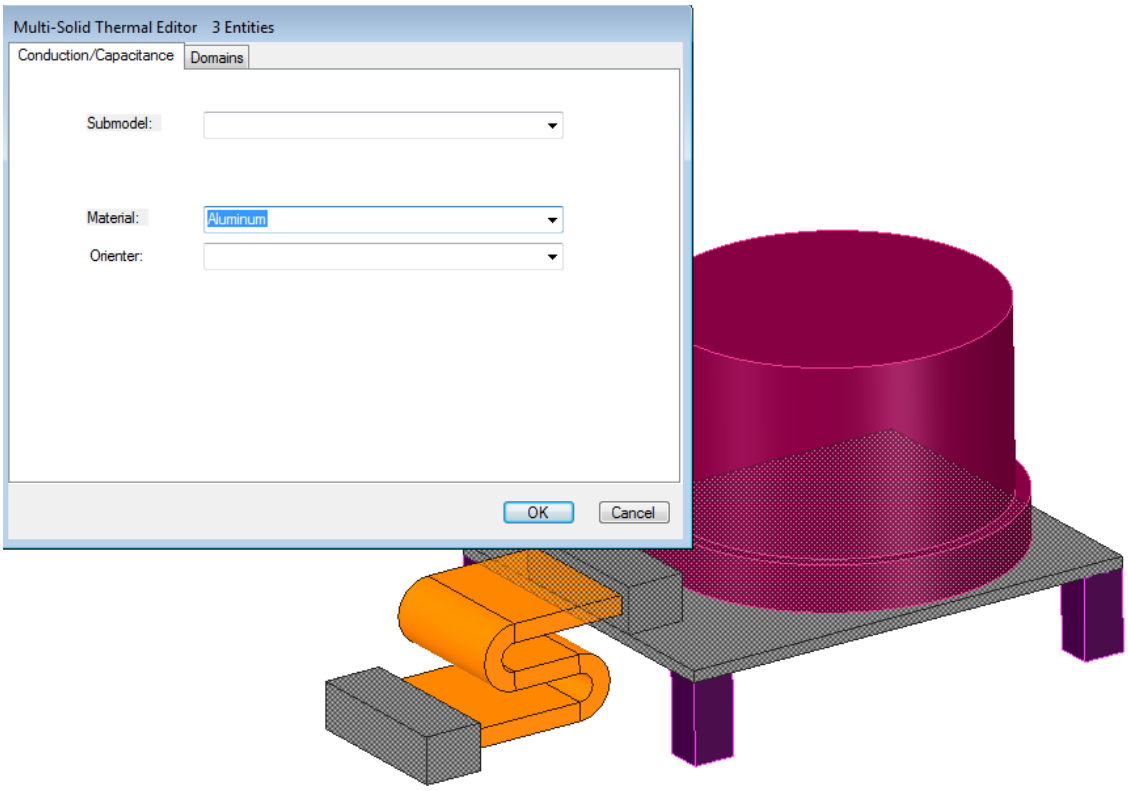
6. Return to TD Direct. Select the solid of the reaction wheel. This can either be done by triple-clicking the reaction wheel itself or by selecting the Solid under the component Reaction_Wheel in the structure tree. With the solid selected, go to the Thermal tab and click on Thermal.



7. On the Conduction/Capacitance tab, use the pull down menu to set the material to **Aluminum**. On the Domains tab, select Add. Type in a new Domain tag named RWheel. This domain tag will be available in Thermal Desktop. Click Ok to add the Domain tag name, and click Ok to close the solid thermal editor window.

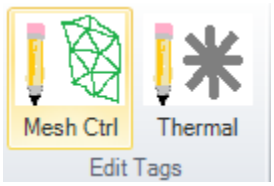


8. Select the solids of the Baseplate, the Strap_Top, and Strap_Bottom. Again, either triple click the solids and hold **Ctrl** to add to the selection, or select the solids from the structure tree. Once they are selected, click on the Thermal button on the Thermal tab. On the Conduction/Capacitance tab, choose **Aluminum** for the material.



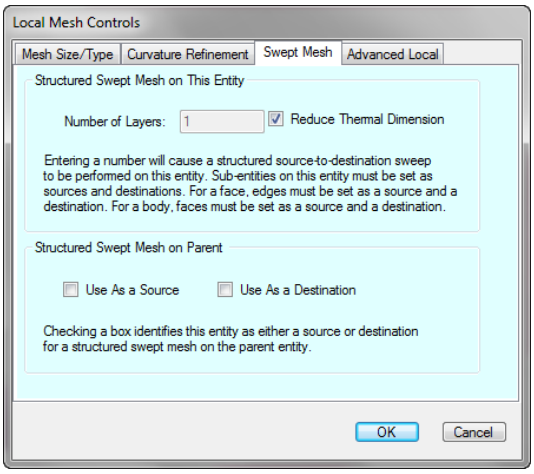
9. Select the four isolators (the purple legs) solids. Using the same method as the previous three steps, set the material to **Isolator**. The thermophysical properties are already defined and are similar to rubber.

10. The conduction through the thickness of the baseplate is insignificant, but replacing it with a surface instead of a solid would require repositioning components to make the correct contacts, and that would involve adjusting the ribbon shape. Instead, the option for a swept mesh with a reduced thermal dimension will be used. The first step is to select the Baseplate solid and click on Mesh Ctrl on the Thermal tab.

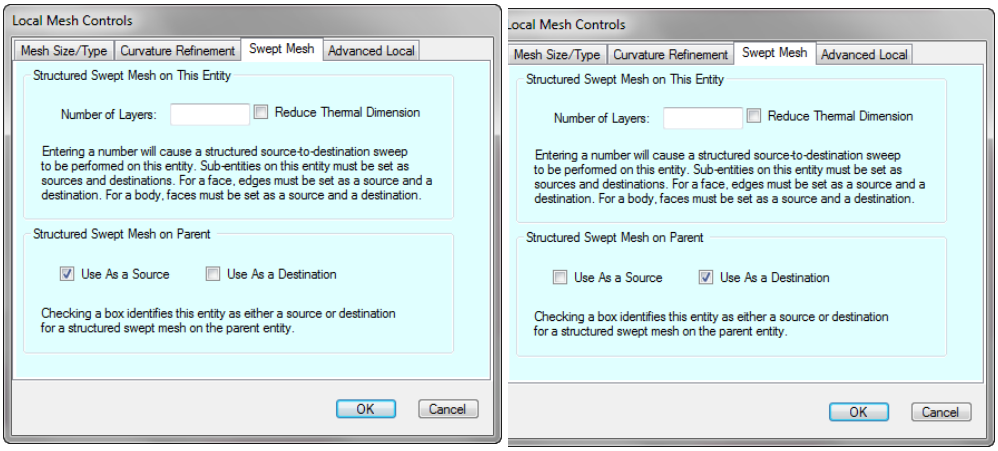


11. On the Swept Mesh tab, there is a box for Number of Layers. If there were a need for high resolution through the thickness of this plate, it could be set to ten, for example, which would create ten layers of the solid through the thickness. This case only needs a single layer. Instead of entering in 1, however, check the box labeled Reduce Thermal Dimension. By checking this box, the number of layers is set to 1, and also the nodes numbers on either side of that layer will be the same.

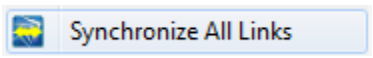
For the purposes of the geometry, it is a thin solid. When the solution goes to SINDA, for the purposes of the solution, it will be a surface. Click Ok to close Local Mesh Controls.

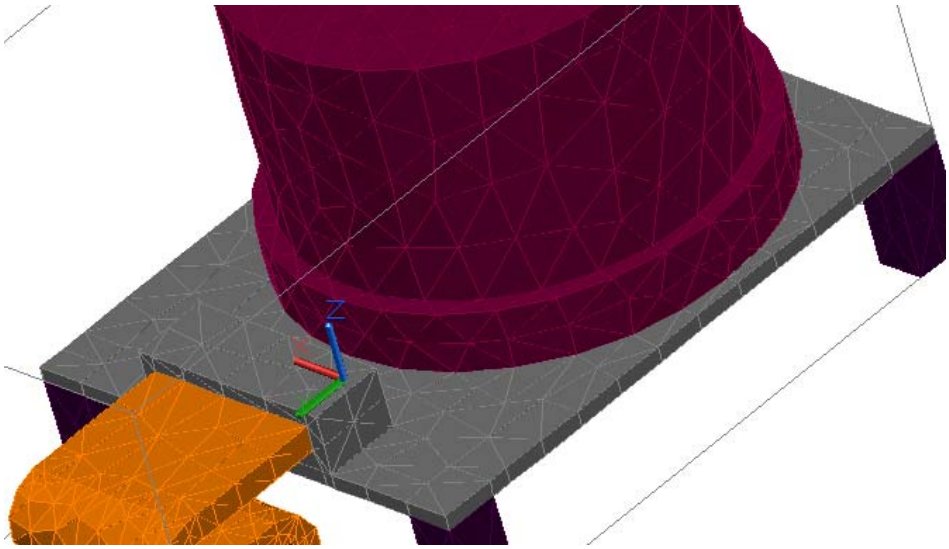

Local Mesh Controls
Mesh Size/Type
Curvature Refinement
Swept Mesh
Advanced Local
Structured Swept Mesh on This Entity
Number of Layers:
1
Reduce Thermal Dimension
Entering a number will cause a structured source-to-destination sweep to be performed on this entity. Sub-entities on this entity must be set as sources and destinations. For a face, edges must be set as a source and a destination. For a body, faces must be set as a source and a destination.
Structured Swept Mesh on Parent
Use As a Source
Use As a Destination
Checking a box identifies this entity as either a source or destination for a structured swept mesh on the parent entity.
OK
Cancel

12. For every swept mesh, a source and destination must be defined. For a solid, the source and destination will be surfaces. When using a swept mesh on a surface, the source and destination will be edges. For the baseplate solid, the source will be the top (the side with the reaction wheel), and the destination will be the bottom (the side with the isolators). Select the top face of the baseplate and click on Mesh Ctrl. Check the Use As a Source box. Click Ok. Do the same with the bottom face, but this time check the box labeled Use As a Destination.

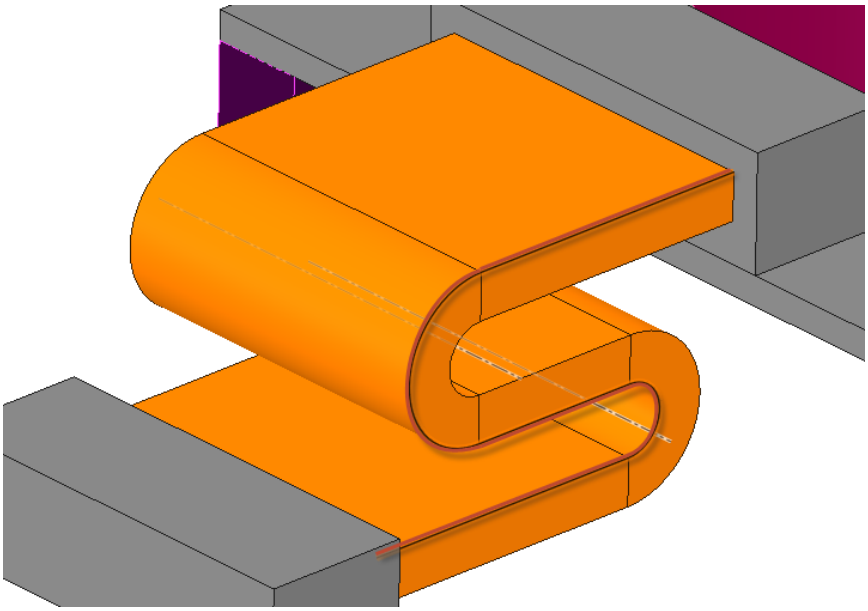

Local Mesh Controls
Mesh Size/Type
Curvature Refinement
Swept Mesh
Advanced Local
Structured Swept Mesh on This Entity
Number of Layers:
Reduce Thermal Dimension
Entering a number will cause a structured source-to-destination sweep to be performed on this entity. Sub-entities on this entity must be set as sources and destinations. For a face, edges must be set as a source and a destination. For a body, faces must be set as a source and a destination.
Structured Swept Mesh on Parent
Use As a Source
Use As a Destination
Checking a box identifies this entity as either a source or destination for a structured swept mesh on the parent entity.
OK
Cancel
Local Mesh Controls
Mesh Size/Type
Curvature Refinement
Swept Mesh
Advanced Local
Structured Swept Mesh on This Entity
Number of Layers:
Reduce Thermal Dimension
Entering a number will cause a structured source-to-destination sweep to be performed on this entity. Sub-entities on this entity must be set as sources and destinations. For a face, edges must be set as a source and a destination. For a body, faces must be set as a source and a destination.
Structured Swept Mesh on Parent
Use As a Source
Use As a Destination
Checking a box identifies this entity as either a source or destination for a structured swept mesh on the parent entity.
OK
Cancel

13. Go to Thermal Desktop. Select *Thermal > TD Direct > Synchronize All Links*. This will synchronize all TD Direct links in the model. Notice how the mesh on the baseplate changed. The mesh on the top and bottom is the same. Further, the imprints of the legs can be seen on the top, and imprint of the top of the ribbon and the reaction wheel can be seen on the bottom.





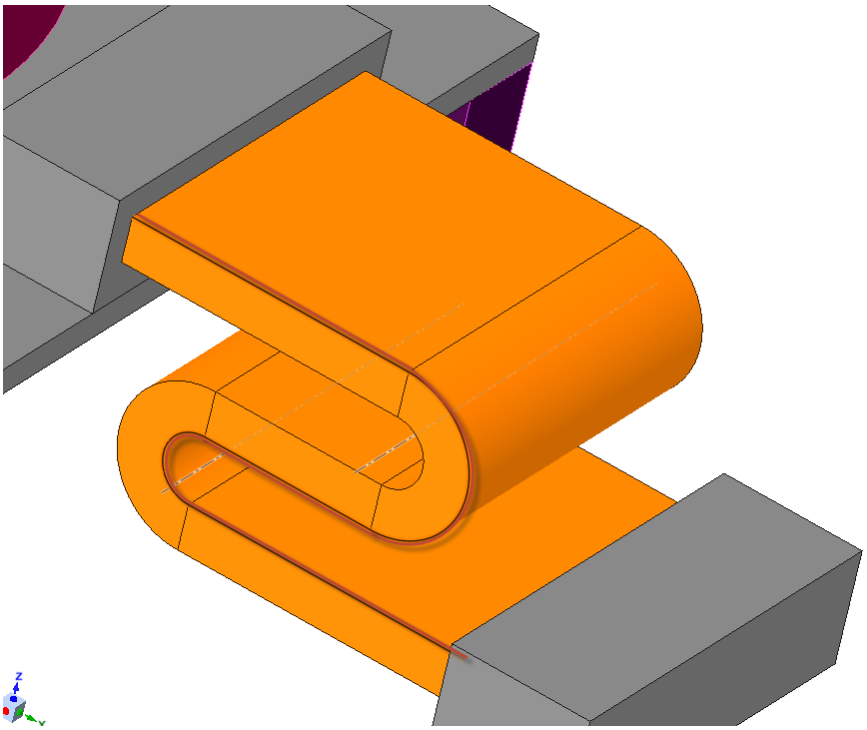
14. After verifying the swept mesh has functioned correctly, return to TD Direct. The next steps will involve using mesh controls on the copper strap. There are many possible methods of creating a swept mesh on the copper. The strap itself has been broken into five solids to follow the restrictions of the swept mesh operation and to keep an orderly mesh. First, select the five edges highlighted below (remember to hold down **Ctrl** to select multiple objects in TD Direct). These five edges make a continuous S. Be sure not to select any faces or solids. On the Thermal tab, select Mesh Ctrl.



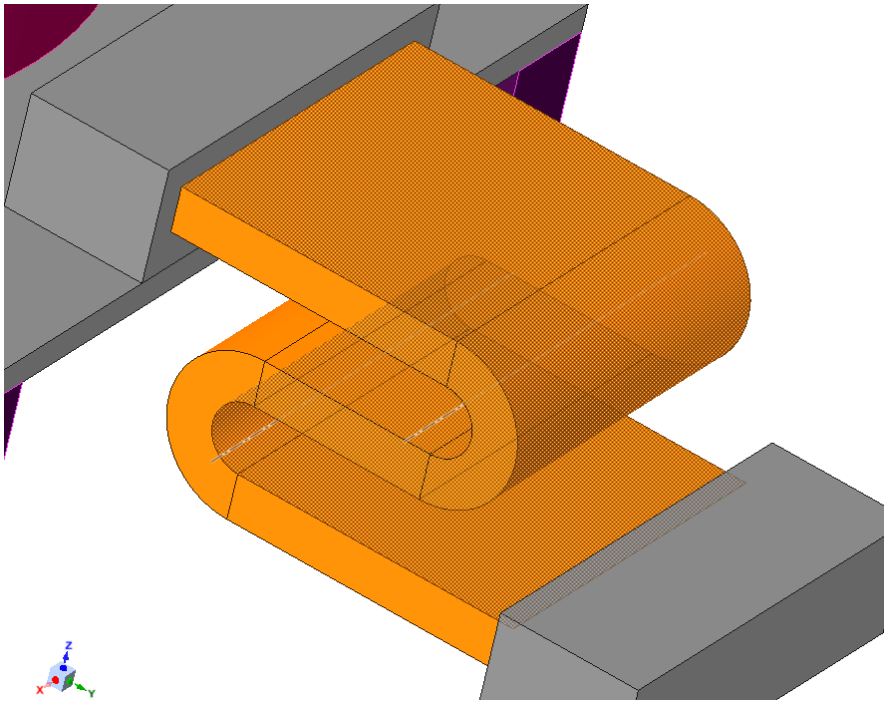
15. On the Swept Mesh tab, check the box Use as a Source. These edges will be swept across a face.

16. Select the edges on the opposite S curve, highlighted below. Click on Mesh Ctrl

with these edges selected, and check the box Use as a Destination.

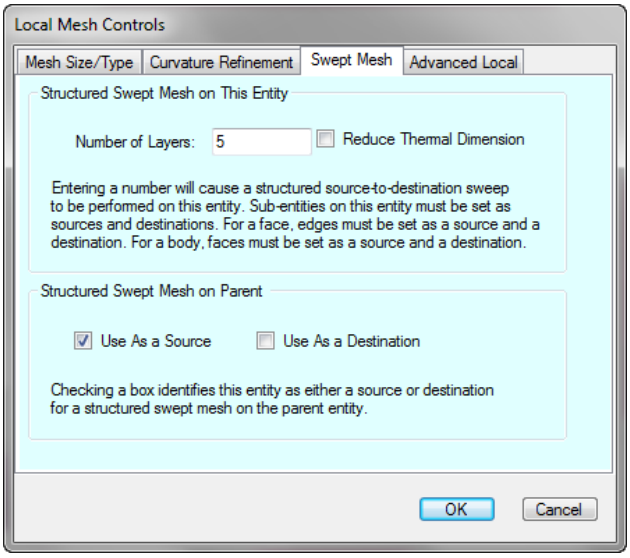


17. Select the top surfaces of the strap (between the previously selected edges) as shown below.

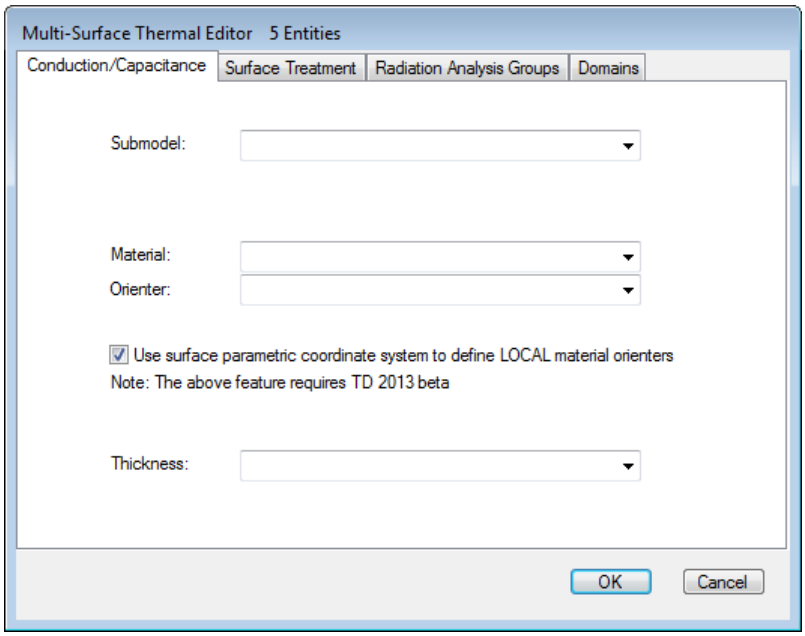


18. Click on Mesh Ctrl. A mesh will be swept from edge to edge across this face. For that swept mesh, enter 5 in the Number of Layers on the Swept Mesh tab. These

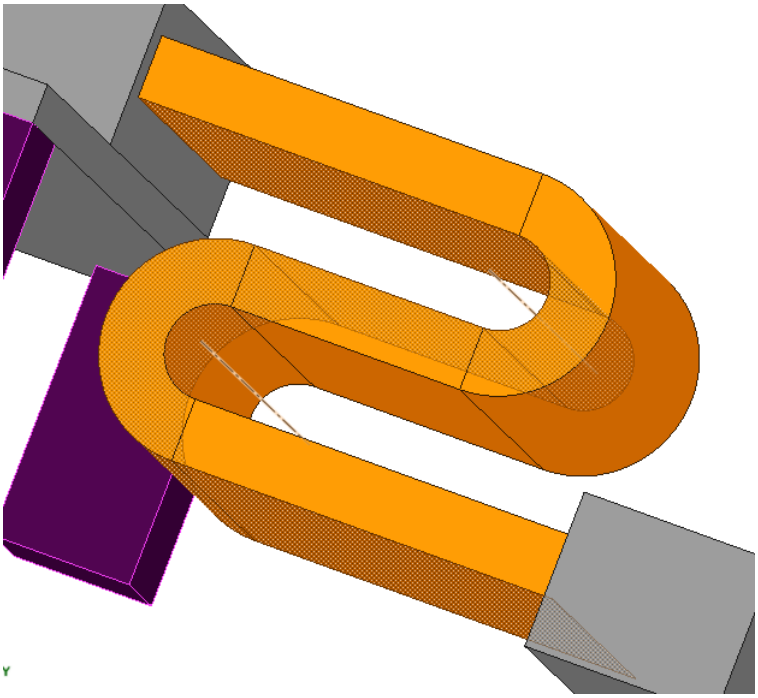
surfaces will also serve as the source faces for a mesh across the solid of the strap, so check the box labeled Use As a Source.


Local Mesh Controls
Mesh Size/Type
Curvature Refinement
Swept Mesh
Advanced Local
Structured Swept Mesh on This Entity
Number of Layers:
5
Reduce Thermal Dimension
Entering a number will cause a structured source-to-destination sweep to be performed on this entity. Sub-entities on this entity must be set as sources and destinations. For a face, edges must be set as a source and a destination. For a body, faces must be set as a source and a destination.
Structured Swept Mesh on Parent
Use As a Source
Use As a Destination
Checking a box identifies this entity as either a source or destination for a structured swept mesh on the parent entity.
OK
Cancel

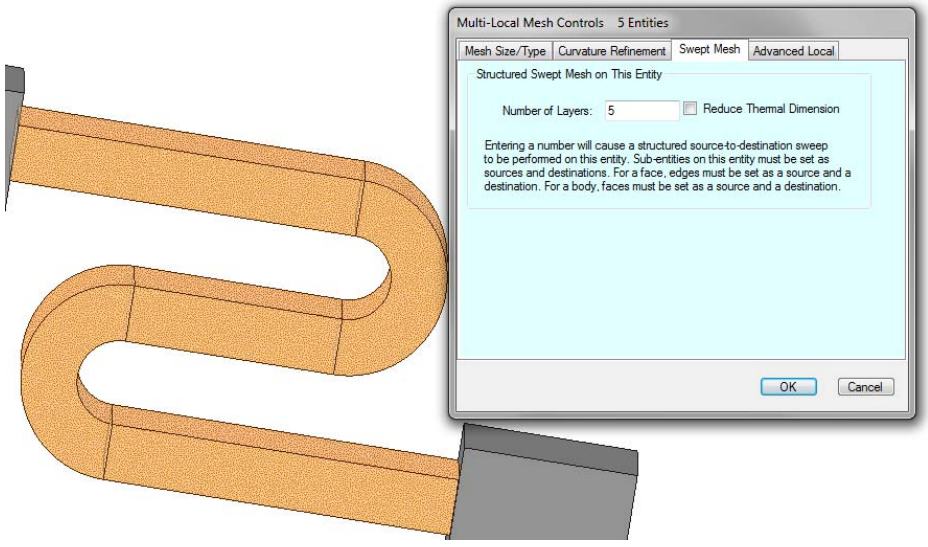
19. With the same surfaces still selected, click on Thermal to edit the tags. The copper strap is being treated as an anisotropic material because the heat transfer from one thin sheet of copper to the one above it or below it is significantly less than the heat transfer along the length of that copper sheet. This surface will define the Z direction for each element of the copper. Check the Box labeled Use surface parametric coordinate system to define LOCAL material orienters.



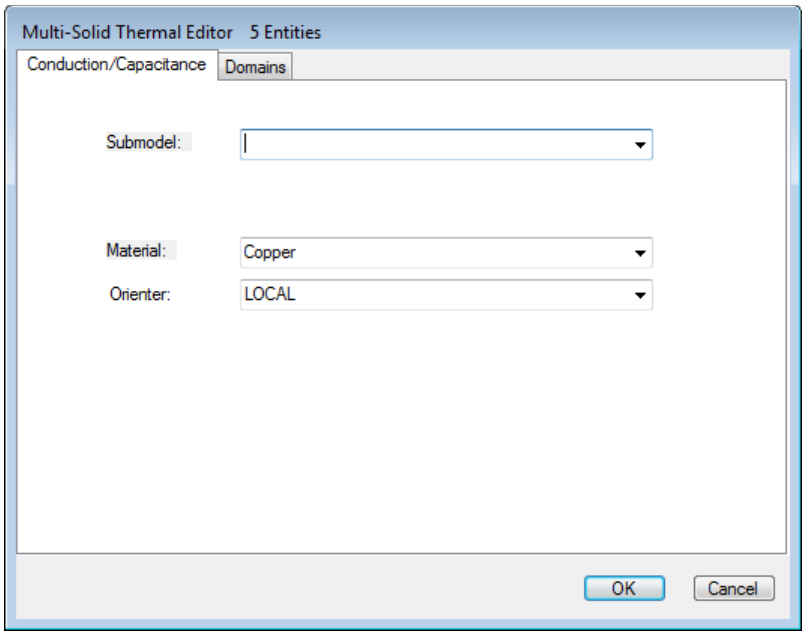
20. Select the bottom surfaces of the strap as shown below. Select Mesh Ctrl, and check the box Use As a Destination.



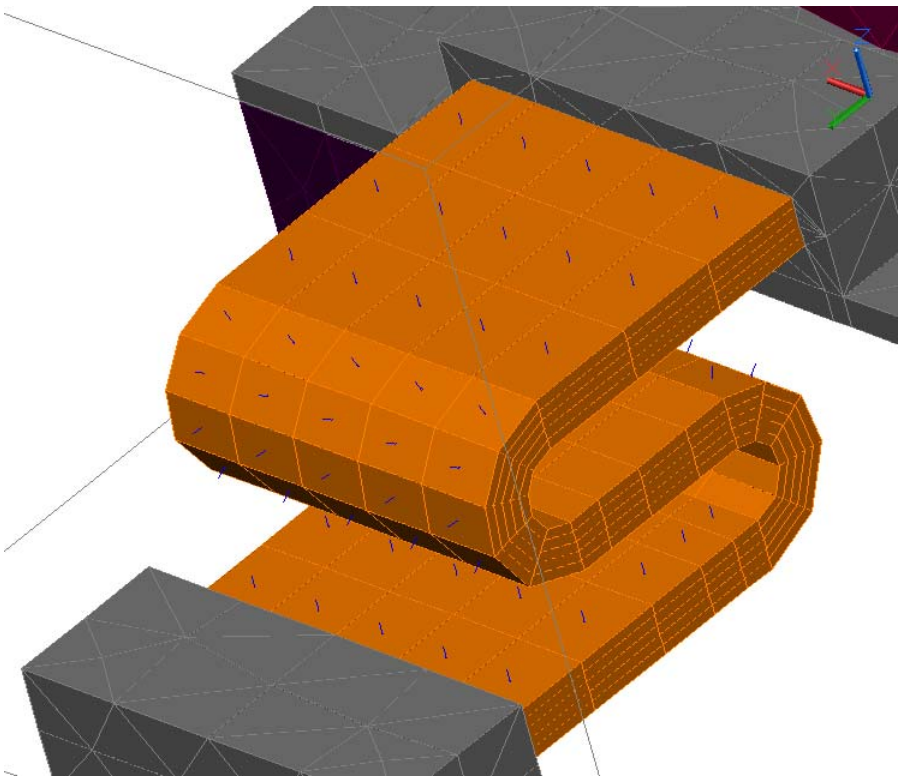
21. Select all five solids of the strap either by triple clicking or using the structure tree (hold down **Ctrl** to make multiple selections). Click on Mesh Ctrl, and set the Number of Layers to 5.


Multi-Local Mesh Controls 5 Entities
Mesh Size/Type
Curvature Refinement
Swept Mesh
Advanced Local
Structured Swept Mesh on This Entity
Number of Layers: 5
Reduce Thermal Dimension
Entering a number will cause a structured source-to-destination sweep to be performed on this entity. Sub-entities on this entity must be set as sources and destinations. For a face, edges must be set as a source and a destination. For a body, faces must be set as a source and a destination.
OK
Cancel

22. With the same five solids selected, click on Thermal to edit the tags. On the Conduction/Capacitance tab, change the material to **Copper** and type **LOCAL** in the Orienter field. Each element in the mesh will use the closest surface defined as an orienter to align the material orienter of that element.

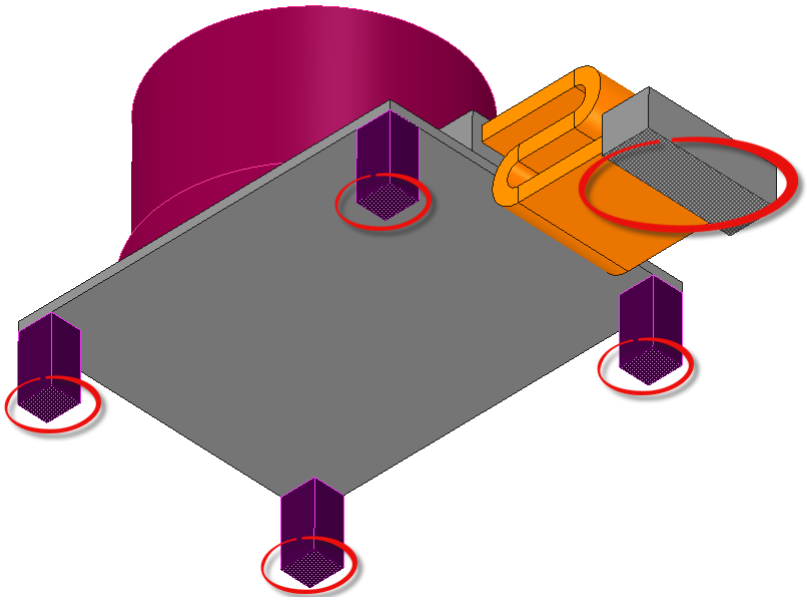


23. After configuring the copper strap, return to Thermal Desktop and verify the mesh is working. Synchronize the mesh by selecting *Thermal > TD Direct > Synchronize All Links*. The mesh should look like the one below. The short blue lines are the Z axis of the material orienter.

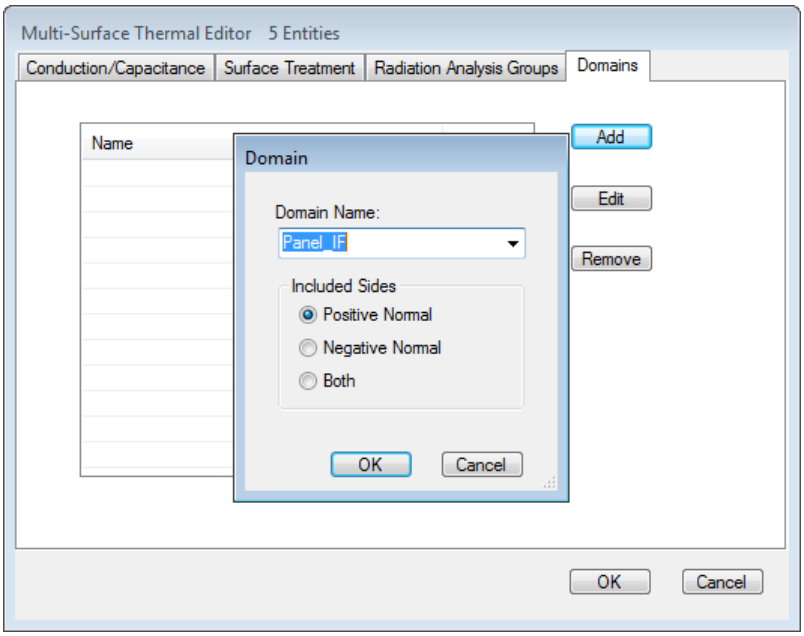


24. With the new mesh verified, all that remains is to apply some boundary conditions. To do this, start by returning to TD Direct. The worst case condition for this analy-

sis will be a where the components are mounted to a panel at 40°C. Select the bottom of the surface of the four isolators and the bottom of the strap mount.



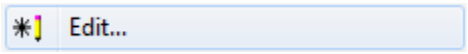
25. Select Thermal to edit the tags. On the Domains tab, Add a domain tag to these surfaces called **Panel_IF**. Use the Positive Normal surfaces. Click Ok to close the Editor window.



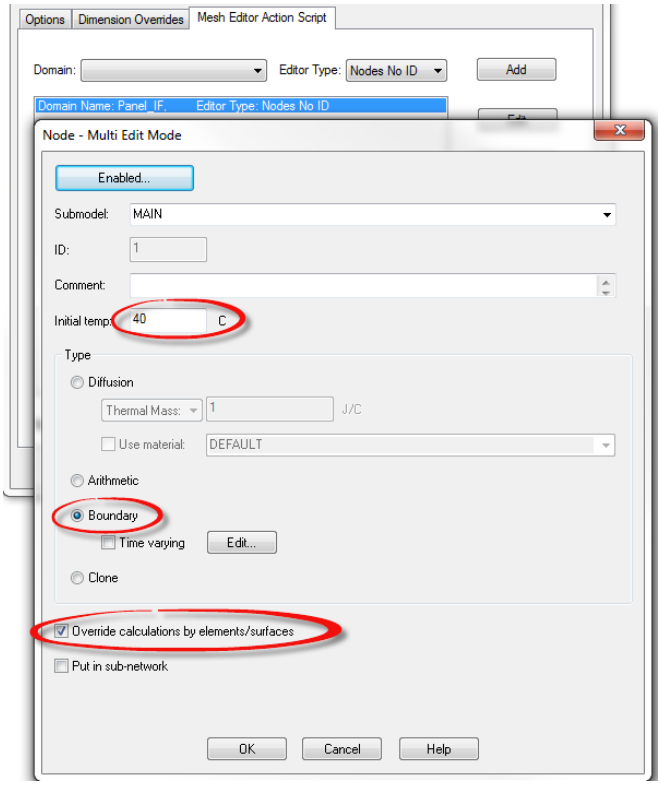
26. Return to Thermal Desktop and synchronize the files again.

27. After the files have synchronized, the new domain tag set will be accessible in Thermal Desktop. Click on the TD Direct Importer, which shows the bounding

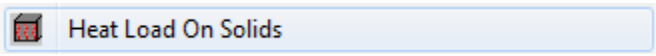
box and the TD Direct document name, and then select *Thermal > Edit*.

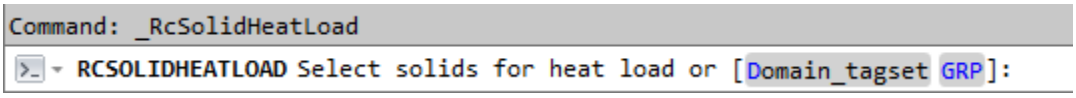


28. On the Mesh Editor Action Script tab, use the pull-down menu to select the Domain **Panel_IF**. Then select the Editor Type **Nodes No ID**. Click Add. This will allow you edit all of the nodes in the Domain Panel_IF. In this case, check the box Override calculations by elements/surfaces, and then use the radio button to select Bounday. Type **40** in the Initial Temperature box to set the temperature of those nodes. Click Ok. Click Synchronize.

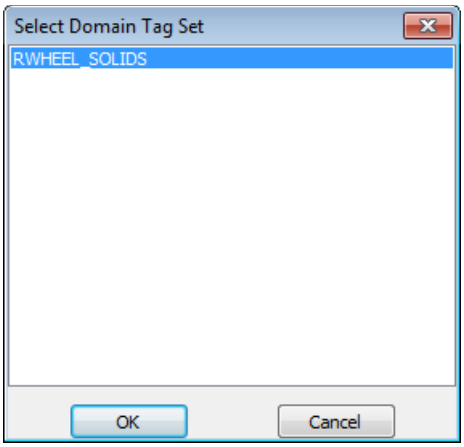


29. Select *Thermal > FD/FEM Networks > Heat Load on Solids*. The command line will prompt you for the solids. In the command line, click on the word "Domain_tagset," or type **D** and press enter.

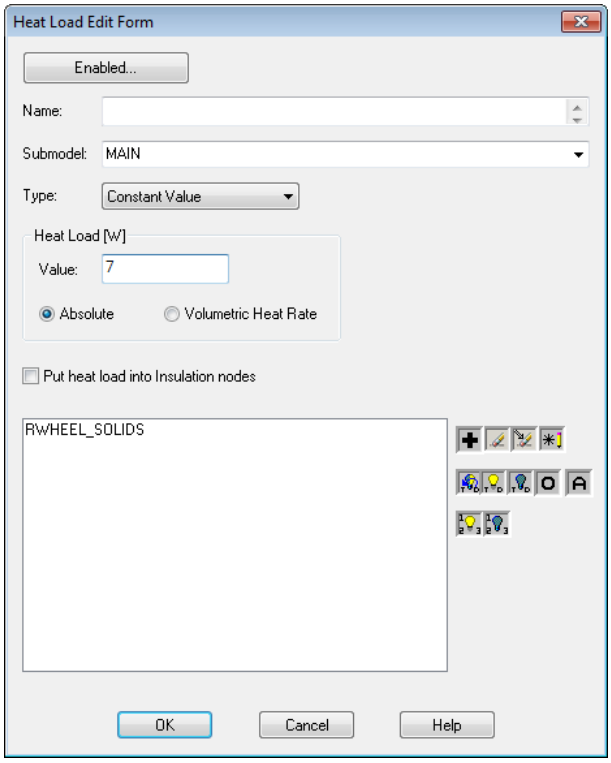




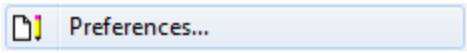
30. The list of available tag sets that represent solids will appear. Select RWHEEL_SOLIDS, which you defined earlier. Click Ok, and type Enter to finish selecting solids.



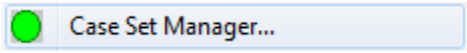
31. This is the heat load applied to the wheel. Change the value to **7** W and click Ok.



32. In order to see things more clearly, select *Thermal > Preferences*. On the Graphics Visibility tab, Deselect User Defined Nodes, Heat Loads, and Material Orienters.



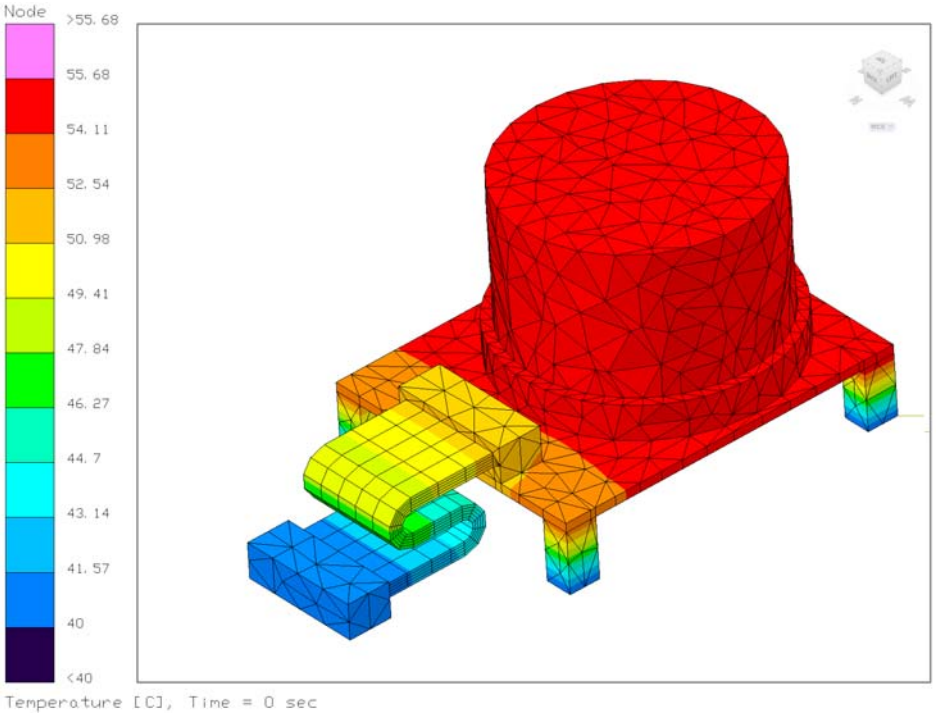
33. The case is ready to run. Select *Thermal > Case Set Manager*. Select Run 1 Selected Case.



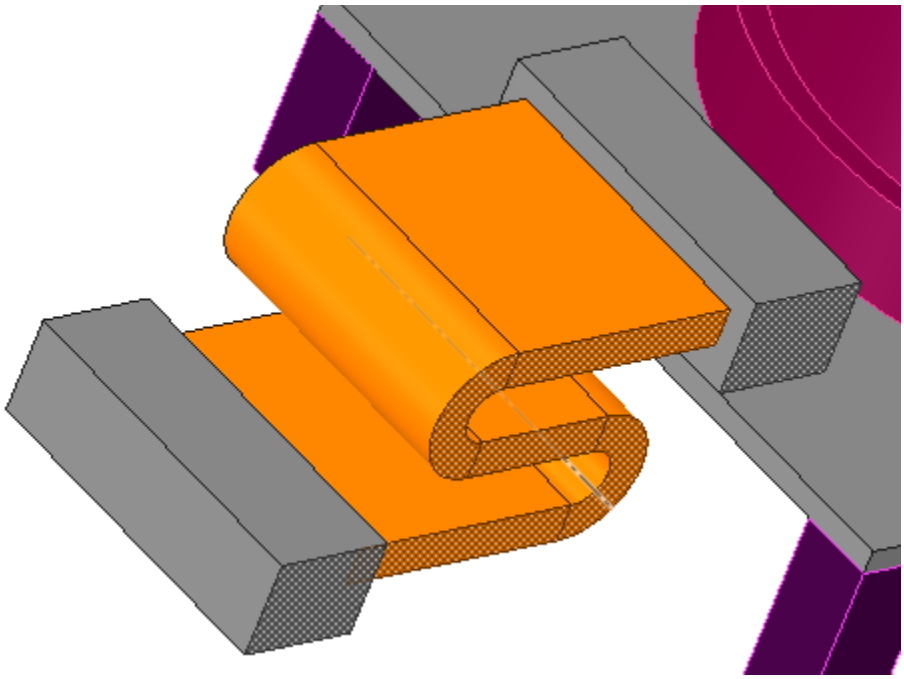
34. A message indicating there are duplicate node numbers may pop up. Select “Allow

The Duplicate Ids to Remain In The Model." The duplicate nodes are from the swept mesh with the reduced thermal dimension on the baseplate.

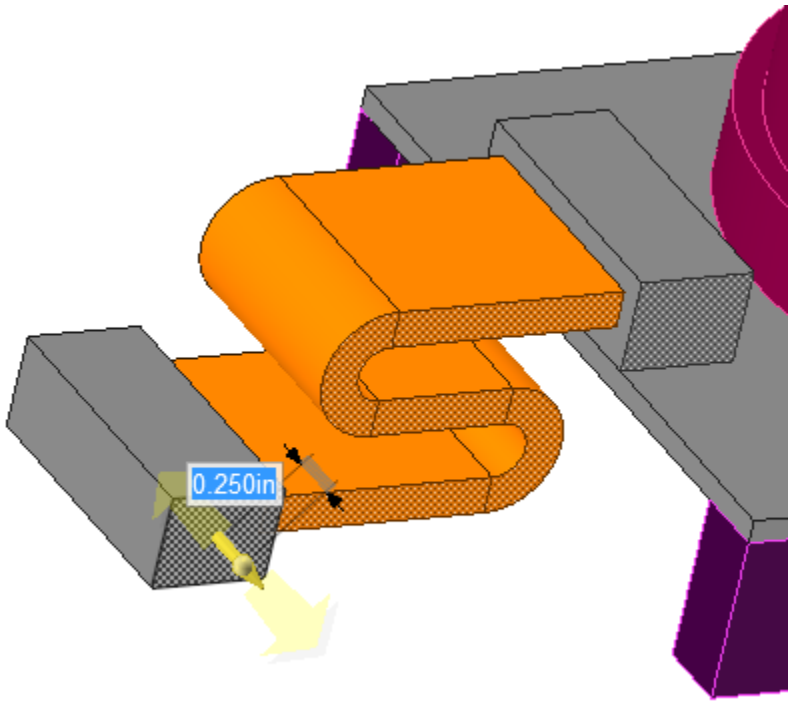
35. The Post-Processing window will appear and show results of this analysis.



36. The allowable temperature limit for the reaction wheel is 65°C. The structural team reported that the copper strap transmitted too much vibration in the frequency of concern. They ask if the ribbon can be made smaller since there is over 9°C extra margin. They hope it can be reduced from 1.5in wide to 1.0in. Return to TD Direct. Select the surfaces of the copper strap and the mounting box on one side as shown below.



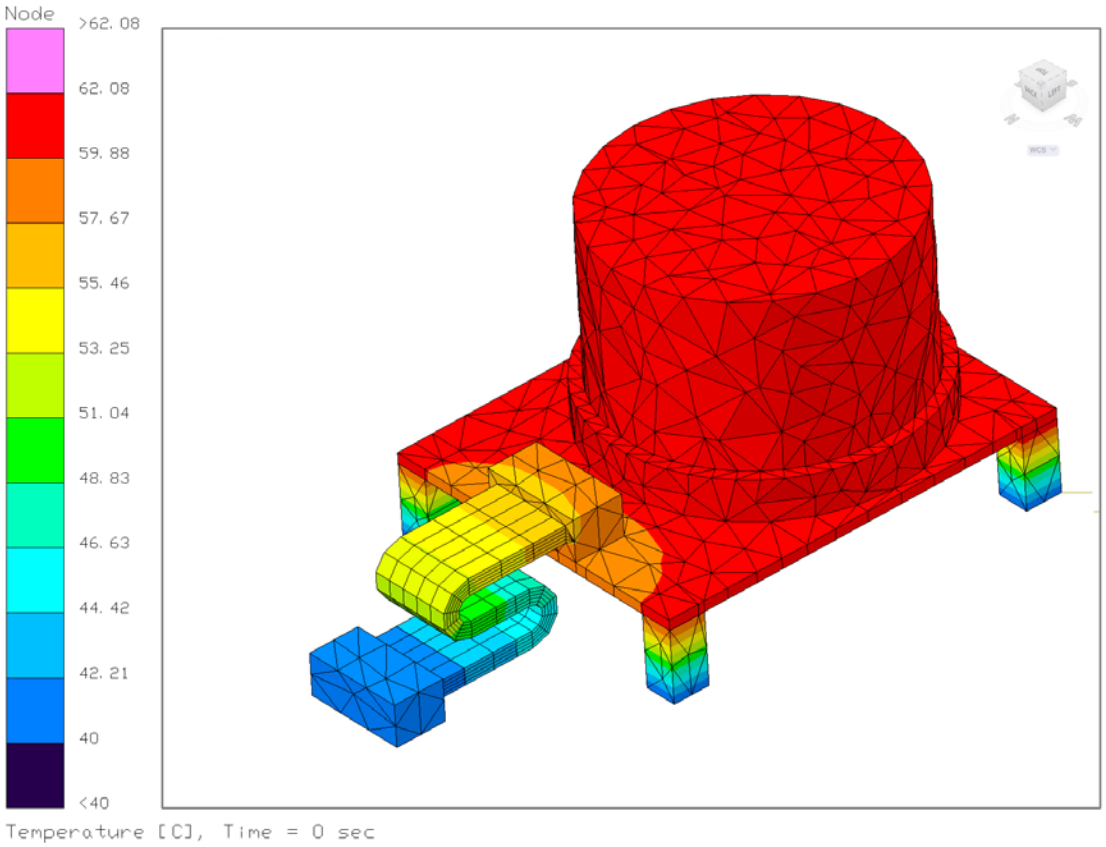
37. Select the Pull tool on the Design tab. Click on the yellow arrow and drag it towards the solid (the direction opposite the way the arrow is pointing). While still holding the left mouse button, press the space bar. Type **0.25** and press **enter**. This will move the selected surfaces back 0.25in. Press Escape to exit the pull command.



38. Repeat the operation on the other side of the copper strap. This will reduce the total width of the ribbon to 1 in and keep it centered on the baseplate.

39. Return to Thermal Desktop. Synchronize the files.

40. Go to the Case Set Manager and rerun the case. Again, allow the duplicate nodes. The new results are shown below. Given the assumptions made, it appears a copper strap 1in wide will meet the requirement of 65°C.



41. Investigate other options on your own. If you have the RadCAD module and are somewhat familiar with it, you can see how radiation affects the analysis. In this case, radiation was neglected. Optical properties for black anodized aluminum, bare copper, and the isolator material are already provided. You may apply these thermal tags in TD Direct and create a radiation task in the Case Set. Set the environment temperature to 40°C to start. Optical properties are applied in TD Direct similarly to the way material properties were defined. You may also consider changing the interface assumption from perfect contact to an interface with a contact resistance. Start with a contact conductance of 2 W/in^2-C.

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